Copy No. 7

DEFENCE R&D DÉFENSE

Ultimate Strength Application Version 1.0

User's Manual

M. Lichodzijewski

T. MacAdam

MARTEC Ltd.,
Suite 400, 1888 Brunswick Street,
Halifax, NS B3J 3J8

DEFENCE RESEARCH ESTABLISHMENT ATLANTIC

Contractor Report

DREA CR 2000-114

March 2000

National Defence Défense nationale

Canada

THIS IS AN UNEDITED REPORT ON SCIENTIFIC OR TECHNICAL WORK CONTRACTED BY THE DEFENCE RESEARCH ESTABLISHMENT ATLANTIC OF DEFENCE R&D CANADA.

THE CONTENTS OF THE REPORT ARE THE RESPONSIBILITY OF THE CONTRACTOR, AND DO NOT NECESSARILY REFLECT THE OFFICIAL POLICIES OF THE DEPARTMENT OF NATIONAL DEFENCE.

PLEASE DIRECT ENQUIRIES TO:

THE DIRECTOR GENERAL
DEFENCE RESEARCH ESTABLISHMENT ATLANTIC
P.O. BOX 1012
DARTMOUTH, NOVA SCOTIA, CANADA
B2Y 3Z7

Ultimate Strength Application Version 1.0

User's Manual

M. Lichodzijewski

T. MacAdam

MARTEC Ltd.,
Suite 400, 1888 Brunswick Street,
Halifax, NS B3J 3J8

Defence Research Establishment Atlantic

Contractor Report

DREA CR 2000-114

March 2000

Authors

M. Lichodzijewski and T. MacAdam

Approved by

Malcolm Smith

Contract Scientific Authority

Approved for release by

Walter E. Ellis

Chair, DREA Document Review Panel

© Her Majesty the Queen as represented by the Minister of National Defence, 2000

ULTIMATE STRENGTH APPLICATION VERSION 1.0
USER'S MANUAL

Martec Report No.: TR-00-14

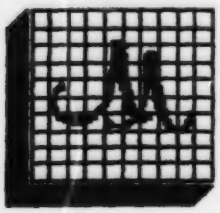
By.

M. Lichodzijewski
T. MacAdam

Prepared for:

Defence Research Establishment Atlantic

Contract No.: W8477-8-0004/001/SV
PWGSC File No.: 005sv.W8477-8-0004



Martec Limited
Advanced Engineering and Research Consultants

400-1888 Brunswick Street
Halifax, NS B3J 3J8
Tel: (902) 425-5101
Fax: (902) 421-1923

101-717 boul St. Joseph
Hull, PQ J8Y 4B6
Tel: (819) 595-3526
Fax: (819) 595-1739

March 2000

ABSTRACT

The document provides instructions for using the Ultimate Strength Analysis Suite (ULTSAS). This programme brings together the UK and Canadian ultimate strength analysis codes in a single software application with a common user interface and a common database. Also provided in the software is a link between ultimate strength analysis tools and the finite-element based ship structural analysis program MGDSA (MAESTRO Graphics – Detail Stress Analysis).

RÉSUMÉ

Le document fournit des instructions sur l'utilisation du programme d'analyse de la contrainte de rupture (ULTSAS). Ce programme regroupe les codes d'analyse de la contrainte de rupture du Royaume-Uni et du Canada afin de constituer une seule application logicielle à interface utilisateur commune et à base de données commune. Le logiciel établit aussi un lien entre les outils d'analyse de la contrainte de rupture et le programme MGDSA d'analyse structurelle des navires à partir d'éléments finis (MAESTRO Graphics - Analyse détaillée de contrainte).

ULTIMATE STRENGTH APPLICATION VERSION 1.0
USER'S MANUAL

DREA CR 2000-114

M. Lichodzijewski and T. MacAdam
Martec Limited

EXECUTIVE SUMMARY

Introduction

Ultimate strength analysis is undertaken to find the bending load that will cause the collapse of a ship hull. Accurate knowledge of this is essential to determine a ship's weight limit and allowable operating conditions. Over the years, specialised analysis codes for evaluating ultimate strength have been developed by defence agencies in several countries. In Canada, the computer code ULTMAT was developed at DREA; in the UK, the NS94 suite of codes was developed at DERA Rosyth. Martec Limited of Halifax Nova Scotia, under contract to DERA Rosyth and DREA, was tasked with bringing together the UK and Canadian ultimate strength analysis codes in a single software application with a common user interface and a common database. A further goal was to provide a link between ultimate strength analysis and the finite-element based ship structural analysis program MGDSA (MAESTRO Graphics – Detail Stress Analysis). The overall objectives were to provide easier access to the analysis tools and to automate the ultimate strength modelling and evaluation process.

Principal Results

This report consists of a User's Manual for the unified ultimate strength application. As a result of the developments undertaken in this work, cross-sectional models can be generated automatically from existing finite element ship models. Once this has occurred, an ultimate strength analysis can be carried out immediately through the application interface. Customisation of the model, data and analysis options is also permitted. Graphical tools are provided for reviewing of results. The net result is a very significant reduction in the time required to carry out ultimate strength modelling and analysis. With the unified application, an ultimate strength assessment can now be carried out in a matter of minutes if a finite element model of the ship already exists. This is a considerable increase in efficiency over previous efforts requiring several hours.

Significance of Results

Automated evaluation of ultimate strength is an essential component of the ongoing Improved Ship Structural Maintenance Management (ISSMM) project. The results of this study directly pertain to how ultimate strength analysis is to be implemented in the ISSMM Software Tool (IST). The automation and improved efficiency that has resulted from this work will mean that ultimate strength analysis will now be accessible to a wider group of users in the CF community. Furthermore, the Navy will be provided with answers to its ultimate strength queries more quickly, and hence a reduced cost.

APPLICATION D'ANALYSE DE LA CONTRAINTE DE RUPTURE, VERSION 1.0 MANUEL DE L'UTILISATEUR

DREA-CR-2000-114

M. Lichodzijewski et T. MacAdam
Martec Limited

SOMMAIRE

Introduction

Une analyse de la contrainte de rupture est entreprise dans le but de déterminer la charge de flexion qui provoquera la rupture d'une coque de navire. La connaissance précise de cette charge est essentielle pour établir le poids limite et les conditions d'exploitation admissibles d'un navire. Avec les années, des organisations de défense de plusieurs pays ont constitué des codes d'analyse spécialisés permettant d'évaluer la contrainte de rupture. Au Canada, le code informatique ULTMAT a été développé au CRDA; au Royaume-Uni, l'ensemble de codes NS94 a été établi au DERA Rosyth. Dans le cadre d'un contrat du DERA Rosyth et du CRDA, Martec Limited d'Halifax (Nouvelle-Écosse) a été chargée de regrouper les codes d'analyse de la contrainte de rupture du Royaume-Uni et du Canada afin de constituer une seule application logicielle à interface utilisateur commune et à base de données commune. Les travaux visaient également à constituer un lien entre l'analyse de la contrainte de rupture et le programme MGDSA d'analyse structurelle des navires à partir d'éléments finis (MAESTRO Graphics - Analyse détaillée de contrainte). Les objectifs globaux consistaient à faciliter l'accès aux outils d'analyse et à automatiser le processus de modélisation et d'évaluation de la contrainte de rupture.

Principaux résultats

Ce rapport comprend un manuel de l'utilisateur portant sur l'application unifiée d'analyse de la contrainte de rupture. Les résultats des travaux effectués permettent de générer automatiquement des modèles de sections transversales à partir des modèles de navires existants à éléments finis. Cela fait, l'interface de l'application peut servir à exécuter immédiatement une analyse de la contrainte de rupture. Il est également possible de personnaliser le modèle, les données et les options d'analyse. Des outils graphiques sont prévus pour l'examen des résultats. Il est ainsi possible de réduire considérablement le temps nécessaire pour effectuer la modélisation et l'analyse de la contrainte de rupture. L'application unifiée permet maintenant d'effectuer une évaluation de la contrainte de rupture en quelques minutes seulement s'il existe déjà un modèle à éléments finis du navire. Cela représente une nette augmentation d'efficacité par rapport aux travaux antérieurs, qui prenaient plusieurs heures.

Importance des résultats

L'évaluation automatisée de la contrainte de rupture constitue un élément essentiel de l'actuel projet de gestion améliorée de la maintenance de la structure des navires (GAMSN). Les résultats de cette étude portent directement sur la façon dont l'analyse de la contrainte de rupture doit être intégrée à l'outil logiciel GAMSN (IST). L'automatisation et l'amélioration d'efficacité qui ont résulté de ces travaux signifient que l'analyse de la contrainte de rupture sera dorénavant accessible à un plus large groupe d'utilisateurs au sein des FC. La Marine recevra donc plus rapidement des réponses à ses questions de contrainte de rupture, ce qui entraînera une réduction des coûts.

TABLE OF CONTENTS

1. INTRODUCTION 1

2. CREATING A NEW ULTIMATE STRENGTH PROJECT 1

2.1. NEW PROJECT WIZARD STEP 1: SELECTING A PROJECT NAME AND LOCATION 1
2.2. NEW PROJECT WIZARD STEP 2: SELECTING INPUT DATA 2
2.3. NEW PROJECT WIZARD STEP 3: SELECTING THE ANALYSIS PROGRAM AND THE LS CURVE DATABASE 3

3. OPENING AN EXISTING PROJECT 5

4. CHANGING PROJECT SETTINGS 5

5. MANIPULATING THE CROSS-SECTION 5

6. SWITCHING BETWEEN FINITE ELEMENT AND STRUCTURAL UNITS MODES 6

7. SELECTING CROSS-SECTION COMPONENTS 7

8. VIEWING PROPERTIES OF CROSS-SECTION COMPONENTS 8

9. MODIFYING PROPERTIES OF CROSS-SECTION COMPONENTS 8

9.1. MODIFICATIONS IN FINITE ELEMENT MODE 9
9.2. MODIFICATIONS IN STRUCTURAL UNITS MODE 11

10. VIEWING GLOBAL CROSS-SECTION INFORMATION 11

11. MAKING GLOBAL MODIFICATIONS TO CROSS-SECTION PROPERTIES 11

12. ACTIVATING/DEACTIVATING A STRUCTURAL COMPONENT 12

13. ASSIGNING A LOAD-SHORTENING CURVE TO A STRUCTURAL UNIT 12

13.1. ASSIGNING A LOAD-SHORTENING CURVE FROM A DATABASE 13
13.2. ASSIGNING THE DEFAULT LOAD-SHORTENING CURVE 13
13.3. GENERATING A LOAD-SHORTENING CURVE BY FEA 14

14. CREATING A USER-DEFINED STRUCTURE 14

15. RUNNING AN ULTIMATE STRENGTH ANALYSIS 15

16. VIEWING SSCP23 RESULTS 16

16.1. GRAPHICAL VIEW OF SSCP23 RESULTS 16
16.2. NUMERICAL VIEW OF SSCP23 RESULTS 16

17. CHANGING THE COLOR SCHEME 17

18. COPYING AND PRINTING CROSS-SECTION BITMAPS 17

19. VIEWING, COPYING AND PRINTING MOMENT-CURVATURE AND INTERACTION CURVES AND DATA 18

TABLE OF FIGURES

FIGURE 1. THE OPENING SCREEN ..... A
FIGURE 2. PROJECT WIZARD STEP 1: PROJECT NAME AND LOCATION ..... A
FIGURE 3. PROJECT WIZARD STEP 2: SOURCE FILES ..... B
FIGURE 4. PROJECT WIZARD STEP 3: ANALYSIS PROGRAM AND LSC DATABASE C
FIGURE 5. PROJECT WIZARD STEP 4: CONFIRMATION C
FIGURE 6. NEWLY GENERATED CROSS-SECTION ..... D
FIGURE 7. CROSS-SECTION INFO DIALOG (GENERAL) ..... D
FIGURE 9. CROSS-SECTION INFO DIALOG (MOMENTS) ..... E
FIGURE 10. CROSS-SECTION INFO DIALOG (STRUCTURE) ..... F
FIGURE 11. CROSS-SECTION INFO DIALOG (LSC PARAMETERS) ..... F
FIGURE 12. CROSS-SECTION INFO DIALOG (SSCP23) ..... G
FIGURE 13. PROJECT SETTINGS DIALOG (PROGRAMS AND DBS) ..... G
FIGURE 14. PROJECT SETTINGS DIALOG (LOAD RANGES) ..... H
FIGURE 15. PROJECT OPTIONS DIALOG (CROSS-SECTION INFO) ..... H
FIGURE 16. PLATE DIALOG (DESCRIPTION) I
FIGURE 17. PLATE DIALOG (MATERIAL) I
FIGURE 18. PLATE DIALOG (DAMAGE) ..... J
FIGURE 19. STIFFENER DIALOG (DESCRIPTION) ..... J
FIGURE 20. STIFFENER DIALOG (MATERIAL) ..... K
FIGURE 21. STIFFENER DIALOG (DAMAGE) L
FIGURE 22. TRANSVERSELY STIFFENED PANEL DIALOG (DESCRIPTION) ..... M
FIGURE 23. TRANSVERSELY STIFFENED PANEL DIALOG (MATERIAL) ..... M
FIGURE 24. TRANSVERSELY STIFFENED PANEL DIALOG (DAMAGE) ..... N
FIGURE 25. TRANSVERSELY STIFFENED PANELS DIALOG (LS-CURVE) ..... N
FIGURE 26. TRANSVERSELY STIFFENED PANEL DIALOG (SSCP23) ..... O
FIGURE 27. LONGITUDINALLY STIFFENED PANEL (DESCRIPTION) ..... P
FIGURE 28. LONGITUDINALLY STIFFENED PANEL (MATERIAL) ..... Q
FIGURE 29. LONGITUDINALLY STIFFENED PANEL (DAMAGE) ..... R
FIGURE 30. LONGITUDINALLY STIFFENED PANEL DIALOG (LS-CURVE) ..... S
FIGURE 31. LONGITUDINALLY STIFFENED PANELS DIALOG (SSCP23) ..... T
FIGURE 32. HARD CORNER DIALOG (DESCRIPTION) ..... T
FIGURE 33. HARD CORNER DIALOG (MATERIAL) ..... U
FIGURE 34. HARD CORNER DIALOG (DAMAGE) ..... U
FIGURE 35. HARD CORNER DIALOG (LS-CURVE) V
FIGURE 36. USER DEFINED STRUCTURE DIALOG (DESCRIPTION) V
FIGURE 37. USER DEFINED STRUCTURE DIALOG (MATERIAL) ..... W
FIGURE 38. USER DEFINED STRUCTURE (DAMAGE) ..... W
FIGURE 39. USER DEFINED STRUCTURE DIALOG (LS-CURVE) X
FIGURE 40. LSPANEL DIALOG (FEA PROGRAM) X
FIGURE 41. LSPANEL DIALOG (GLOBAL CONTROL PARAMETERS AND BOUNDARY CONDITIONS) ..... Y
FIGURE 42. LSPANEL DIALOG (MESHING PARAMETERS) ..... Y
FIGURE 43. LS-CURVE GENERATION DIALOG (BEFORE FEA) ..... Z
FIGURE 44. LS-CURVE GENERATION DIALOG (AFTER FEA) ..... Z
FIGURE 45. FINITE ELEMENT VIEWING MODE ..... AA
FIGURE 46. STRUCTURAL UNITS VIEWING MODE ..... AA
FIGURE 47. GRAPHICAL VIEW OF SSCP23 RESULTS ..... BB
FIGURE 48. RESULTS DIALOG (PLOT) ..... BB
FIGURE 49. RESULTS DIALOG (DATA) CC
FIGURE 50. GLOBAL PROPERTIES DIALOG CC
FIGURE 51. MATERIAL DEFINITION DIALOG ..... DD

FIGURE 52. STIFFENER DEFINITION DIALOG ..... DD
FIGURE 53. COLOR SCHEME DIALOG ..... EE
FIGURE 54. THE FILE MENU. ..... EE
FIGURE 55. THE EDIT MENU ..... FF
FIGURE 56. THE VIEW MENU ..... FF
FIGURE 57. THE ANALYSIS MENU. ..... FF
FIGURE 58. THE RESULTS MENU ..... FF
FIGURE 59. THE OPTIONS MENU ..... GG
FIGURE 60. THE HELP MENU ..... GG

1. INTRODUCTION

This document provides instructions for using the Ultimate Strength Analysis Suite (ULTSAS). It assumes that the reader has an understanding of the concepts associated with ultimate strength analysis.

The reader may wish to read this document in its entirety to understand the operation of the ULTSAS, or jump directly to the subsection that covers a specific task.

2. CREATING A NEW ULTIMATE STRENGTH PROJECT

The ULTSAS, like most modern applications, is project oriented. In order to use the ULTSAS, the user must either create a new ULTSAS project or open an existing ULTSAS project. Creating a new ULTSAS project is discussed here, with the next section discussing opening an existing ULTSAS project.

A new ULTSAS project can be created in one of two ways: either by selecting the New item from the File menu (Figure 54), or by clicking on the New button on the toolbar (Figure 1). Both methods launch the New Project Wizard, which takes the user through the three steps necessary to create a new project. Each step corresponds to a screen on the New Project Wizard and is described below.

2.1. New Project Wizard Step 1: Selecting a Project Name and Location

The first step in creating a new ultimate strength project requires the user to specify a project name, which will be used to name project files and the directory that will contain them.

This step also requires the user to provide a root directory where the new project directory will be created. The default root directory is 'Projects' found at the same level as UltimateStrength.exe.

Figure 2 shows the property page where the user enters this information.

For a description of Ultimate Strength project files refer to Appendix A of *Incorporation of UK NS94 Ultimate Strength Analysis In MGDSA Part II: Software Considerations*.

2.2. New Project Wizard Step 2: Selecting Input Data

The second step is the selection of a cross-section.

In this step, the user must first indicate whether he or she will be importing a cross-section defined in MGDSA or DIGFEM format and then specify the required input files.

For MGDSA, the user must provide the path of a valid '.usd' file.

For DIGFEM, he or she must provide the names of valid '.tbs', '.tta', '.tmp', and 'mfl' files. Once these file names have been provided, he or she must specify a particular cross-section location.

Figure 3 shows the property page where the user enters this information.

Appendix C *of Incorporation of UK NS94 Ultimate Strength Analysis In MGDSA Part II: Software Considerations* contains a description of the MGDSA '.usd' file format.

2.3. New Project Wizard Step 3: Selecting the Analysis Program and the LS Curve Database

The third step involves the selection of the analysis type, the analysis program, and the load-shortening curve database. (The user may accept the default values and move on, since these setting may be changed later; see section *Changing Project Settings*)

The Ultimate Strength application supports two analysis types. The first results in a moment-curvature curve and the second in a bi-axial interaction curve.

After deciding on the analysis type, the user must select the program that is to be used for the ultimate strength analysis. The actual list of analysis programs that may be used for ultimate strength analysis is restricted by the choice of analysis type as not all programs support all analysis types as shown in Table 2.1.

Analysis Type	**Supported by Programs**
Moment-Curvature Curve	NS94A, NS94B, NS94C, ULTMAT
Bi-Axial Interaction Curve	NS94B, NS94D

Table 2.1: Analysis Type Support

If the user selects NS94A, NS94B, or ULTMAT, then he or she can also choose a load-shortening curve database that will be used for generating curves for the cross-section's structural units. Otherwise, the only load-shortening curve database that can be used is the NS94D database. Table 2.2 shows which programs can use which load-shortening curve databases.

Load-Shortening Curve Database	Applicable to Programs
NS94D	NS94A, NS94B, NS94C, NS94D, ULTMAT
VAST (End-To-End Model)	NS94A, NS94B, ULTMAT
VAST (Center-To-Center Model)	NS94A, NS94B, ULTMAT
Special LS-Curve	NS94A, NS94B

Table 2.2: LS-Curve Database Usage

There are three load-shortening databases that may be used. The first is the VAST database containing curves produced using the 'end-to-end' model. The second is the VAST database containing curves produced using the 'center-to-center' model. The third is the NS94D database.

If the user selects one of the VAST databases, then the user also has an option of selecting one of nine sets of 16 curves. Each set contains curves that were generated for a particular combination of stiffener area to total area ratio and maximum plate imperfection.

Figure 3 shows the property page where the user specifies the analysis program and load-shortening curve databases.

A more complete description of the VAST and NS94D databases is available in *Incorporation of UK NS94 Ultimate Strength Analysis In MGDSA Part II: Software Considerations*. A description of the ultimate strength analysis programs and their capabilities can be found in *Incorporation of UK NS94 Ultimate Strength Analysis In MGDSA Part I: Engineering Considerations*.

3. OPENING AN EXISTING PROJECT

The user may wish to open an existing ULTSAS project rather than create a new one. In order to open an existing Ultimate Strength project, the user must select the Open item from the File menu, or click on the Open button on the toolbar. This will display a file browser dialog where the user should specify the desired project file name. Ultimate Strength project files end with the extension '.ULTSAS'.

Figure 6 shows an existing project after having been opened by the user. Figure 54 show the File menu with the Open menu item.

4. CHANGING PROJECT SETTINGS

Once a project has been created, its settings can be accessed through the Project Settings item on the Options menu.

Some project settings are initially set when the project is created. These include the settings chosen on the Analysis Program and Databases page of the New Project Wizard (Figure 13). Other settings include the load range and curvature ratio data (Figure 14) and general cross-section information such as frame location and orientation and the measurement units used (Figure 15).

5. MANIPULATING THE CROSS-SECTION

Once a cross-section is loaded into the ULTSAS, it can be manipulated in a variety of ways using the mouse. Holding down the left mouse button and dragging the mouse translates ("pans") the cross-section in the display window. Holding down the right mouse button and dragging the mouse zooms the cross-section. Dragging in a

downward direction magnifies the cross-section, and dragging in an upward direction shrinks the cross-section (Note that it is possible to zoom in "too close" to the cross-section, such that the image becomes inverted and mouse movements appear to work backwards. This behavior is caused by algorithms internal to the HOOD system. To remedy this situation, "zoom in" until the cross-section image returns to its proper orientation.).

The cross-section can be re-positioned to the center of the window at any time by selecting Fit To Window from the View menu (this can also be used to remedy the inverted rendering situation described above).

The mouse is also used to select components in the cross section. This is described in detail in the section *Selecting Cross-Section Components*.

6. SWITCHING BETWEEN FINITE ELEMENT AND STRUCTURAL UNITS MODES

As described in *Incorporation of UK NS94 Ultimate Strength Analysis In MGDSA Part II: Software Considerations*, the cross section can be viewed in one of two modes: Finite Element mode (Figure 45) and Structural Units mode (Figure 46). In Finite Element mode, each stiffener or plate component is selectable individually. It is in this mode that component properties such as materials and damage can be viewed and modified.

In the other mode, Structural Units mode, individual plate and stiffener components are joined to form stiffened-panels and hard corners. It is in this mode that load-shortening curves are associated with components and can be viewed and modified. This mode allows the user to view component descriptions, material and damage

properties and SSCP23. Note, however, that this mode does not permit modifications to material and damage properties.

The user can switch between Finite Element and Structural Units modes by selecting either Stiffeners-Plates or Structural Units from the View menu, respectively. The current mode is indicated by a checkmark to the left side of the menu item. Alternatively, the corresponding toolbar buttons can also be used to toggle between these two modes.

7. SELECTING CROSS-SECTION COMPONENTS

Once a project is loaded into the ULTSAS and a cross-section is visible, its structural components can be selected. A user selects components to view and/or modify their properties (i.e. description, material, or damage properties). Components can be selected either individually or in groups. If changes are made to the properties of a component that was selected individually, then those changes are applied only to that component. If changes are made to the properties of a group of components, then those changes apply only to those components that are part of that group.

Selecting individual components is achieved by double-clicking (using the left mouse button) on any component within the cross section. The selected component changes color to indicate it is selected, and its corresponding dialog is displayed. Double-clicking on another component simply causes that component to become the current selection and updates the dialog with its data. Refer to the section on modifying component properties to see how to modify the properties of a selected component.

Selecting groups of components is achieved by entering the group selection mode by choosing the type of component from the Edit menu. The possible choices are plates and stiffeners in Finite Element mode and structural units in Structural Units

mode. Once the user enters group selection mode, selection is done by right-clicking on one or more components in the cross-section. Components change color to reflect that they are selected. Once all desired components are selected, the user exits group selection mode by again choosing the component type entry in the Edit menu (it should have a checkmark on its left side). Exiting group selection mode displays the common property page dialog for the selected components. Any changes made to the entries in this dialog will apply to all the components in the selection group.

8. VIEWING PROPERTIES OF CROSS-SECTION COMPONENTS

The properties of individual or aggregate cross-section components can be viewed by selecting the components in the cross section and displaying their associated dialogs. Refer to the section *Selecting Cross-Section Components* for instructions on how to do this.

In Finite Element mode, the description, material properties and damage properties can be viewed for plate and stiffener components (Figures 16 through 21) . In Structural Units mode, the description, material properties, damage properties, load shortening curve and SSCP23 data can be viewed for both transversely and longitudinally stiffened panels (Figures 22 through 31). The description, material properties, damage properties and load shortening curve can be viewed for hard corners and user-defined structures (Figures 32 through 39).

9. MODIFYING PROPERTIES OF CROSS-SECTION COMPONENTS

The properties of individual or aggregate cross-section components can be modified by selecting the components in the cross section and displaying their associated property page dialog screens. Refer to the section *Selecting Cross-Section Components* for

instructions on how to do this. This section outlines the modifications that can be made to cross-section components in both Finite Element and Structural Units modes.

9.1. Modifications in Finite Element Mode

Definitions of individual plates and stiffeners can be modified in the Finite Element mode only (see section *Switching Between Finite Element and Structural Units Modes*). In order to modify a particular plate or stiffener definition, that plate or stiffener must be selected and its property page dialog must be visible (see section *Selecting Cross-Section Components*).

Each plate and stiffener dialog contains three property pages. The first provides a description that includes plate or stiffener dimensions. The second shows the associated material and the third show the associated damage. The following list describe these pages in more detail:

- ***Plate Description Page*** – (Figure 16) here the length and thickness of the plate can be modified. The width of a plate cannot be modified as it is determined by the locations of the end points of the plate.
- ***Plate Material Page*** – (Figure 17) here the plate's material can be modified. This is done by selecting a material from a drop down list. If the user wishes, he/she can create a new material by clicking on the 'Define a New Material' button. This displays a dialog that allows the user to enter the name of the new material (must be unique across the material database) and the material's properties (i.e. modulus of elasticity, Poisson's ratio and yield stress). If the user creates a new material, it is added to the material database.
- ***Plate Damage Page*** – (Figure 18) here damage can be applied to the plate. Two types of damage are supported: corrosion and distortion. Corrosion is specified as either a percentage or absolute measure. There are also three subtypes for

corrosion: uniform, sinusoidal or edge. Each type is specified by selecting the appropriate type from the drop down list and entering the data in the entry fields below it. All three types can be applied simultaneously (although each must be entered separately).

- ***Stiffener Description Page*** – (Figure 19)here resides the definition of the stiffener. This includes such things as the stiffener's location, orientation, dimensions and moments of inertia. The stiffener can be modified by selecting another stiffener from the drop down list or by creating a new stiffener definition. The latter is done by clicking on the 'Define a New Stiffener' button. If the user wishes to create a new stiffener, they must specify a unique name for it, as well as its dimensional values. The stiffener's area, offset and moments of inertia can be automatically calculated after the dimensions are known by hitting the Calculate From Dimensions button If the user creates a new stiffener, it is added to the stiffener database.
- ***Stiffener Material Page*** – (Figure 20) here the stiffener's material can be modified. This page is essentially identical to the Plate Material Page (described above).
- ***Stiffener Damage Page*** – (Figure 21)here damage can be applied to a stiffener. The damage for the stiffener is very similar to that for the plate except that the corrosion damage is applied to the web and flange portions of the stiffener separately.

Once satisfied with his or her changes, the user can accept them by pressing the Apply button or the OK button. Both these buttons modify the plate or stiffener definition, except that the Apply button will keep the property sheet open while the OK button will close it. If the user does not want the changes to take effect, then he or she can cancel the operation and close the property sheet by pressing the Cancel button.

If the user has applied changes to a plate or stiffener, then the associated structural units (in Structural Units mode) will be deactivated, indicating a change in their definitions that requires the user to assign to each a new load-shortening curve. Section

Assigning a Load-Shortening Curve to a Structural Unit describes how to assign a new load-shortening curve to a unit.

9.2. Modifications in Structural Units Mode

In the Structural Units mode, the load-shortening curve can be modified, if the currently selected ultimate strength analysis program can support such changes. See the section *Assigning a Load-Shortening Curve to a Structural Unit* for a description of modifying load shortening curves for cross-section components. This mode does not permit changes to component definitions or material or damage properties.

10. VIEWING GLOBAL CROSS-SECTION INFORMATION

Various data that apply to the cross-section as a whole can be viewed by selecting 'Section Info' from the View menu (shown in Figure 56). These include cross-section location and orientation, units of measurement, cross-section dimensions, moments of inertia, bending moments, component statistics, load-shortening curve parameters and SSCP23 data. The property pages of the dialog used to show this information can be seen in Figures 7 through 12.

11. MAKING GLOBAL MODIFICATIONS TO CROSS-SECTION PROPERTIES

The user has the ability to modify global cross-section properties by selecting 'Global Properties' from the Edit menu (shown in Figure 55). Doing so displays the 'Global Properties' dialog box (Figure 50), from which such properties as cross-section length and material properties can be modified. Clicking OK on this dialog saves any modifications the user has made to the global cross-section properties.

12. ACTIVATING/DEACTIVATING A STRUCTURAL COMPONENT

A structural component can be either active or inactive in relation to its use during the ultimate strength analysis. An active structural component will be taken into account during the analysis, whereas an inactive component is disregarded by the analyzing program (effectively, it is as if there were no component located at the site of an inactive component).

A component's activity can be toggled in its property page dialog when the cross-section is in Structural Units mode (see section *Switching Between Finite Element and Structural Units Modes*). This dialog is displayed when a component or group of components is selected (see section Selecting Cross-Section Components). If an individual component is being activated or inactivated, the Disabled checkbox is checked or unchecked on the Description tab of its property page dialog (see Figures 22, 27, 32, and 36). If a group of components are being activated or inactivated, the Disabled checkbox is checked or unchecked on the Load-Shortening Curve tab of the group property page dialog.

13. ASSIGNING A LOAD-SHORTENING CURVE TO A STRUCTURAL UNIT

From time to time, the user may wish to change the load-shortening curve associated with a structural unit, or assign a load-shortening curve to either a user-defined structure or a component that recently had its properties modified (e.g. damage applied). The user can only change the load-shortening curve assignment if the program to be used for the ultimate strength analysis is not NS94B or NS94D. Otherwise, he/she is constrained to using the assigned load-shortening curve.

Load shortening curves can only be viewed/assigned to components in the Structural Units mode. See section *Switching Between Finite Element and Structural Units Modes* for instructions on entering this mode.

If a user wishes to revert the status of assigned load shortening curves back to their original assignments, as when the project was created, he or she can do so at any time by selecting Re-Initialize LSCs from the Edit menu (as shown in Figure 55).

13.1. Assigning a Load-Shortening Curve from a Database

One way the user can assign a new load-shortening curve to a component is by assigning it from the load-shortening curve database currently being used by the project. In order to do this, the user need only select a new curve from the drop down list on the LS Curve tab of the component's property page dialog (Figures 25, 30, 35, and 39). Once the new load shortening curve is chosen, the user can apply it to the component by clicking either Apply or OK.

13.2. Assigning the Default Load-Shortening Curve

If a new load-shortening curve has been assigned to a component, the original load-shortening curve can be reassigned to the component through the LS Curve tab of the component's property page dialog (Figures 25, 30, 35, and 39). This is done by clicking on the 'Use Default' button. Clicking on the 'Use Default' button automatically applies the change to the component; the user does not need to click the Apply or OK button.

13.3. Generating a Load-Shortening Curve by FEA

For longitudinally stiffened panels, the user may also generate load shortening curves using a FE analysis. This can be done by clicking the Generate Curve button on the LS Curve tab of the longitudinally stiffened panel's dialog (Figures 25, 30, 35, and 39). Clicking this button displays the LSPanel Input Parameters dialog (Figures 40 through 42). This dialog allows the user to setup the FEA by selecting such parameters as the FEA program to be used, the meshing parameters, global control parameters and boundary conditions as described in LS Panel documentation.

Once these parameters are set, the user clicks OK to advance to the LS Curve Generation control window (Figures 43 and 44). Here the user has the opportunity to manually modify the input data to the FEA program, run the FEA, and view the results. If the results are unsatisfactory, the user can re-edit the input data and re-run the FEA. Once the results are satisfactory, the user can assign a name to the curve and accept the curve (click the Accept curve button). The status of the FEA is displayed in the upper right portion of the LS Curve Generation control window at all times.

14. CREATING A USER-DEFINED STRUCTURE

In Structural Units mode, the user is able to group components to form user-defined structures. This is achieved in much the same way as groups of components are selected. In Structural Units mode, the user must first select New UDS from the Edit menu (shown in Figure 55). The components that are to form the user-defined structure are then selected by right-clicking on them in the cross-section. Selected components change color to indicate that they are selected. Once all the components that will form the user-defined structure are selected, the user again chooses New UDS from the Edit menu. The user will be prompted whether or not to proceed, and if

he/she chooses Yes, the user-defined structure will be created and displayed in a distinct color in the cross section.

The user-defined structure is initially set to inactive when it is created to indicate that the user must assign to it a load shortening curve. See section *Assigning a Load-Shortening Curve to a Structural Unit* to see how this is done.

15. RUNNING AN ULTIMATE STRENGTH ANALYSIS

Once the user has made the desired modifications to the cross-section, he/she can run the ultimate strength analysis on it by selecting the Run menu item in the Analysis menu (Figure 57). The analysis will be run using the current settings for the project (see section *Changing Project Settings*).

There are three stages to running an analysis. The status of the ULTSAS as it goes through these stages is shown in the status bar (lower left corner of main window). The first stage comprises generating input for the program that will perform the analysis. The second stage consists of the actual analysis. During this stage, a dialog is displayed to allow the user to cancel the analysis. The third stage consists of the ULTSAS reading and formatting the results of the analysis for display.

Upon completing of the third stage, the results are automatically displayed to the user by the ULTSAS. This display consists of the curvature – bending moment plot and data listing (Figures 48 and 49). See section *Viewing, Copying and Printing Moment-Curvature and Interaction Curves and Data* for instructions on how to print/copy this result data. The result display is also available following an analysis by selecting Plots and Tables from the Results menu.

16. VIEWING SSCP23 RESULTS

SSCP23 results can be viewed graphically and numerically. These results are described in more detail in *Incorporation of UK NS94 Ultimate Strength Analysis In MGDSA Part II: Software Considerations*.

16.1. Graphical View of SSCP23 Results

In the graphical view, a value of a particular limit state associated with a particular structural unit is shown through that unit's color. The mapping between limit state values and colors is provided through a color ramp visible at the side of the main window. Since some limit state values are not relevant to some structural units, those structural units are colored with gray.

In order to graphically view the color coding for a particular limit state, the user must select the limit state that he or she wants to see from the 'Results:SSCP23 Limit States' menu. On occasions when no applied bending moments were specified, the menu options dealing with comparisons to applied stress are disabled.

Figure 47 shows a sample SSCP23 limit state result.

16.2. Numerical View of SSCP23 Results

In order to view the actual SSCP23 limit state values associated with a particular structural unit, the user must select that structural unit, open its property dialog, and click on the SSCP23 tab (Figures 26 and 31). (See sections *Selecting Cross-Section Components* and *Viewing Properties of Cross-Section Components*.).

To see the ranges of limit state values as they exist throughout the cross-section, the user must open the "Cross-Section Information" dialog and click on the SSCP23 tab (Figure 12). (See section *Viewing Global Cross-Section Information*.)

17. CHANGING THE COLOR SCHEME

The user is able to change the color scheme for the cross-section display window. This is done by selecting the Color Scheme item from the Options menu (Figure 59). The color scheme modification dialog lists each item of the cross-section display and shows its current display color (Figure 53). The color can be changed for each item by clicking the Change button next to the desired item. The display can be reset to the default color scheme at any time by clicking on the Default button.

18. COPYING AND PRINTING CROSS-SECTION BITMAPS

The user may wish to either print or copy an image of the cross-section display to another program.

To print the cross-section, the user must first copy the cross section bitmap into another program like Microsoft Word and print it using that program's facilities.

To copy an image of the cross-section to the Windows Clipboard, select Copy from the Edit menu (or click the copy button on the toolbar). A bitmap image of the current display window will be copied to the clipboard, which then can be pasted into most other Windows programs. Note that the bitmap is a copy of the current cross-section view; if the cross-section is translated and/or zoomed, it will appear translated and/or zoomed in the bitmap copy.

19. VIEWING, COPYING AND PRINTING MOMENT-CURVATURE AND INTERACTION CURVES AND DATA

Following an ultimate strength analysis, the moment-curvature plot and associated data is automatically displayed for the user (these results can also be manually displayed following an ultimate strength analysis by choosing the Plots and Tables item from the Results menu).

The moment-curvature plot can be manipulated in a number of ways. The user can zoom in on a portion of the plot by clicking the Zoom button on the toolbar and dragging the mouse over the area of the plot he/she wishes to zoom. The zoom can be disabled by again clicking on the Zoom button on the toolbar. The X and Y gridlines can be hidden/displayed by clicking on the gridline buttons on the toolbar. The plot title and axis labels can be modified by clicking on the Edit Titles button on the toolbar. The remaining buttons (Change Text Fonts, Tools, Change Chart Options) enable the user to further refine the detailed look of the plot.

The moment-curvature plot can be printed by clicking on the Print button on the toolbar. This prints the plot on the default Windows printer, scaled to fit a typical 8.5x11 inch sheet of paper (aligned "portrait"). A bitmap image of the plot can also be copied to the Windows Clipboard by clicking on the Copy button on the toolbar. This image can be pasted into most other Windows programs.

The moment-curvature plot data is displayed on the Data tab of the Results window. The data can be viewed in this window or copied to the Windows Clipboard. To copy the data to the Windows Clipboard, simply select the portion of data you wish to copy by dragging the mouse over it, then type Ctrl+C. This data can also be pasted into most Windows programs.

APPENDIX A

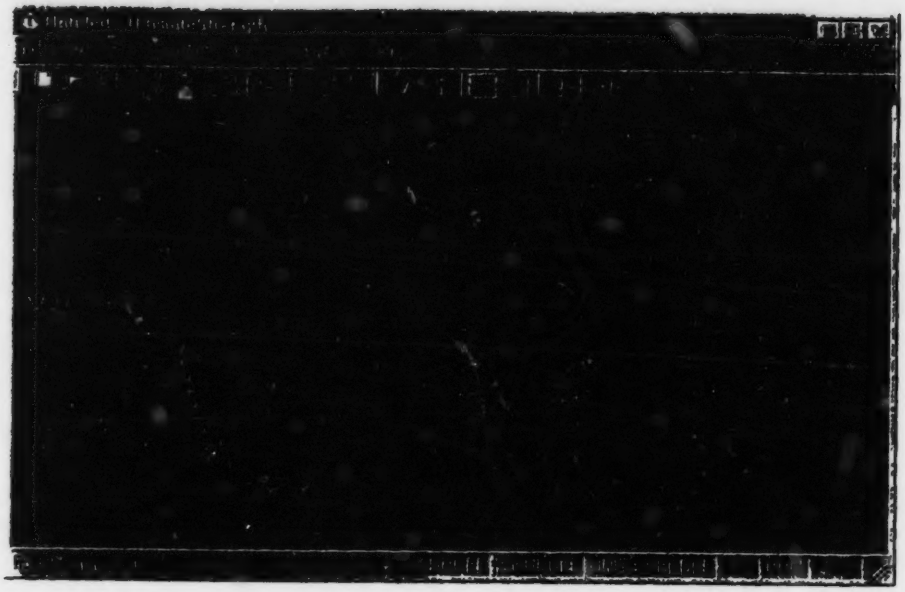


Figure 1. The Opening Screen

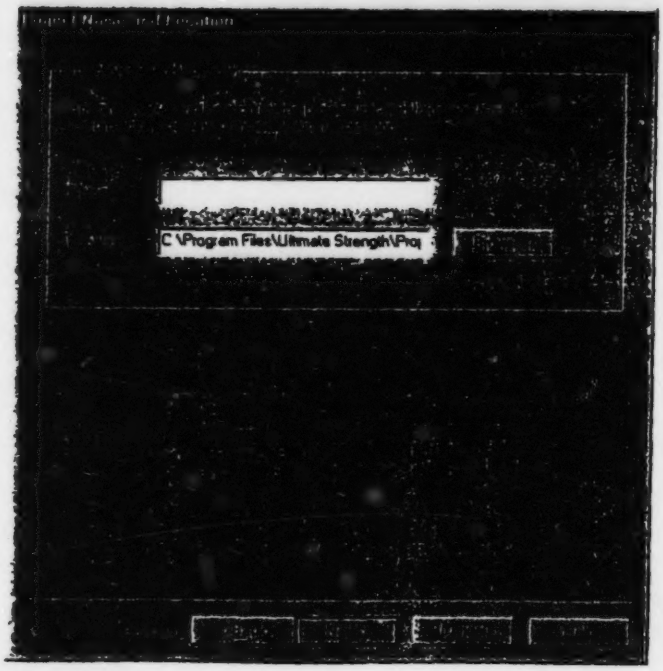


Figure 2. Project Wizard Step 1: Project Name and Location

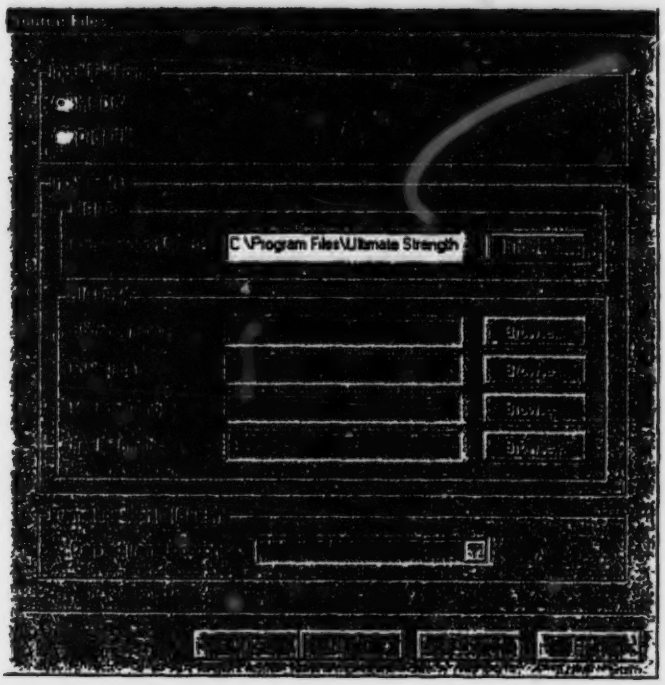


Figure 3. Project Wizard Step 2: Source Files

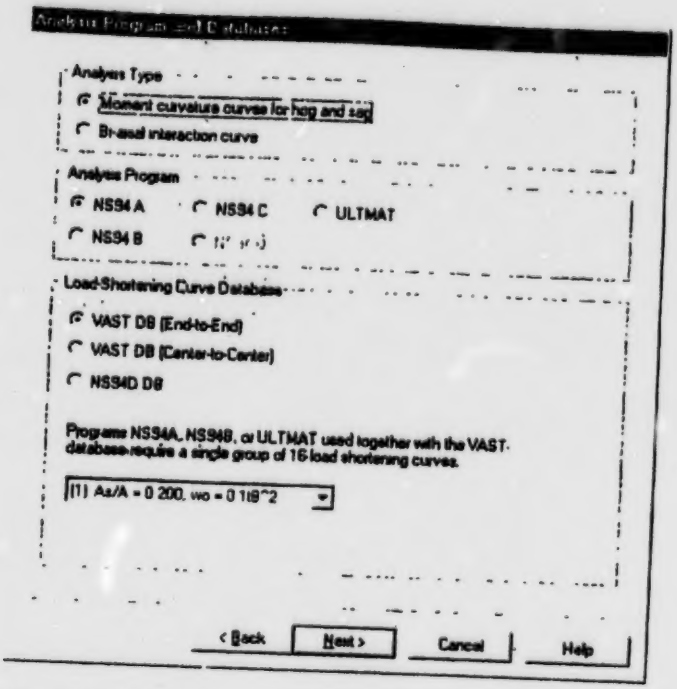


Figure 4. Project Wizard Step 3: Analysis Program and LSC Database

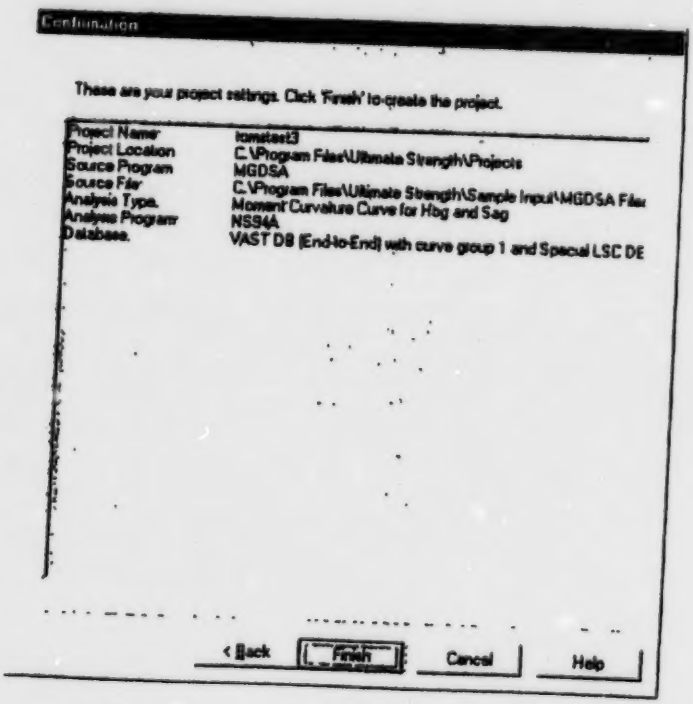


Figure 5. Project Wizard Step 4: Confirmation

d

Figure 6. Newly Generated Cross-Section

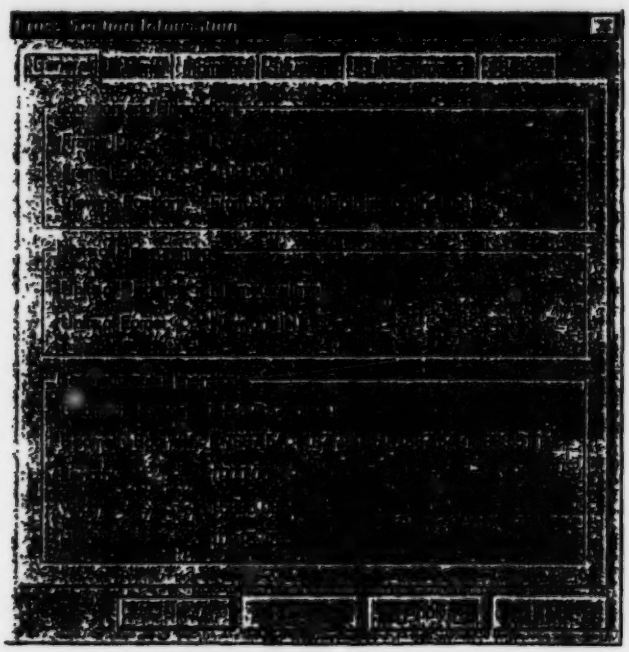


Figure 7. Cross-section Info Dialog (General)

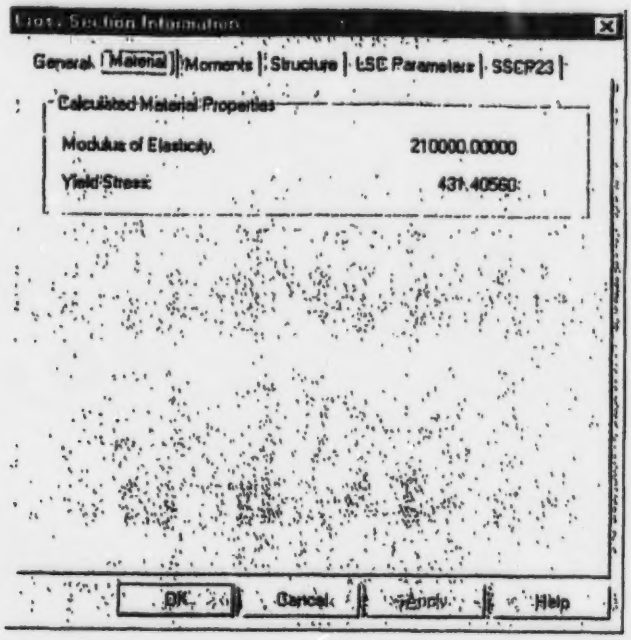


Figure 8. Cross-section Info Dialog (Material)

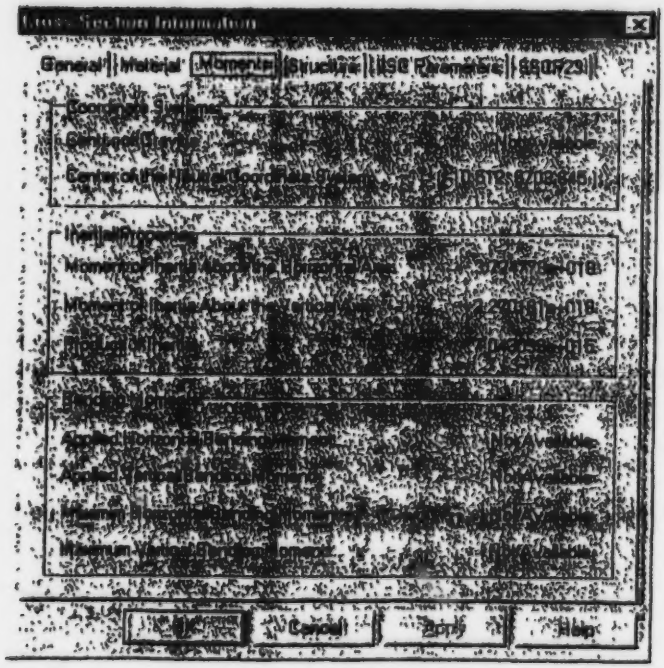


Figure 9. Cross-section Info Dialog (Moments)

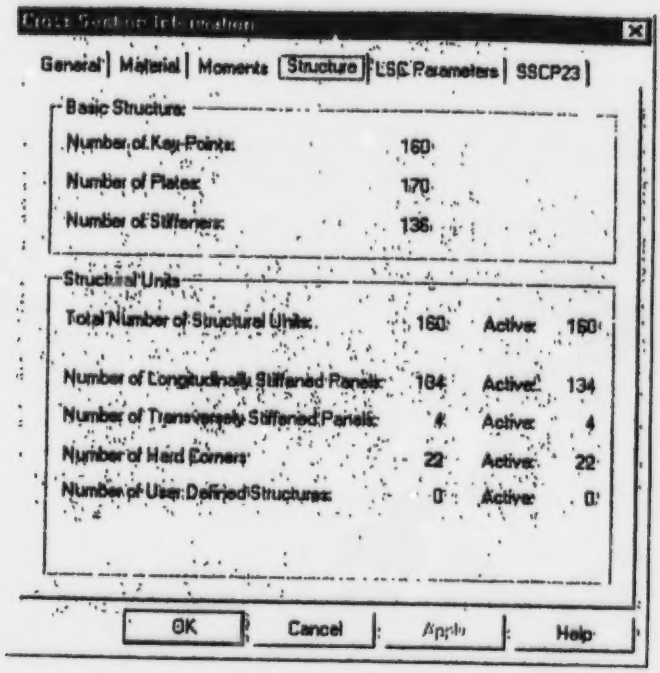


Figure 10. Cross-section Info Dialog (Structure)

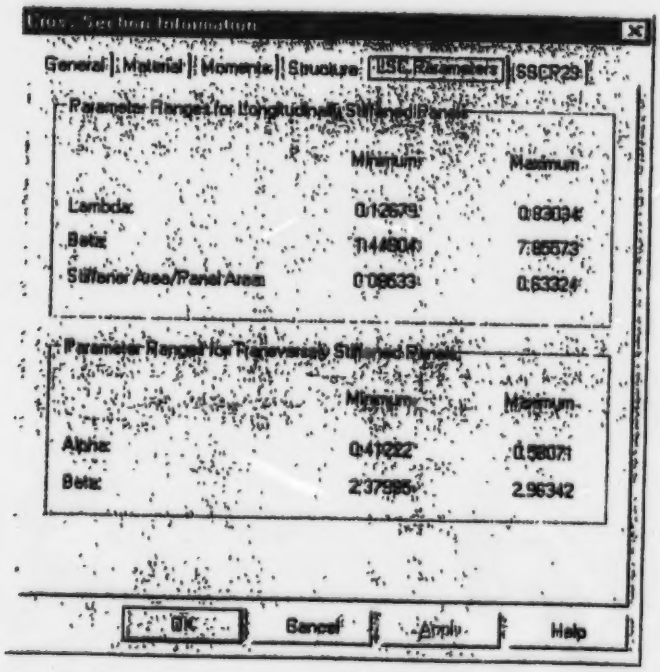


Figure 11. Cross-section Info Dialog (LSC Parameters)

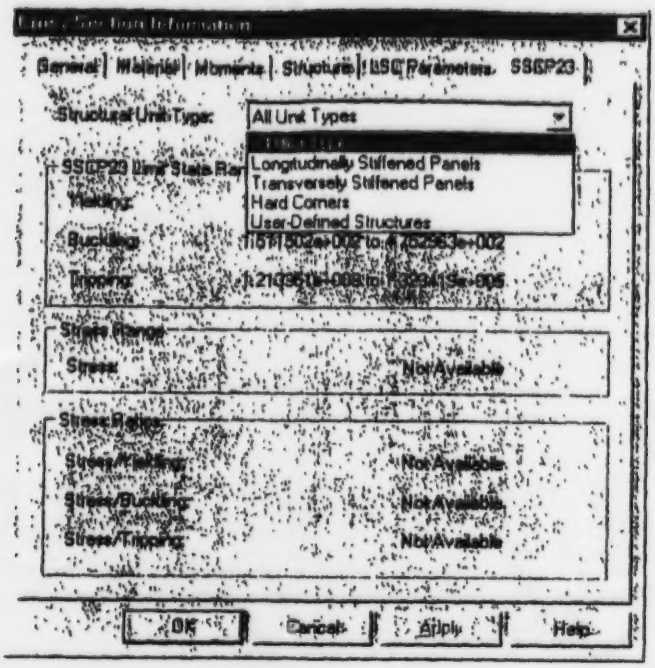


Figure 12. Cross-section Info Dialog (SSCP23)

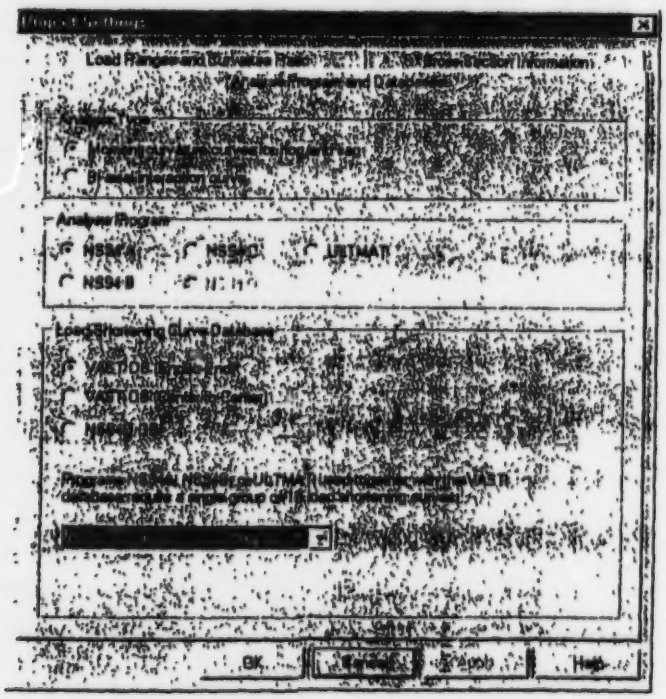


Figure 13. Project Settings Dialog (Programs and DBs)

h

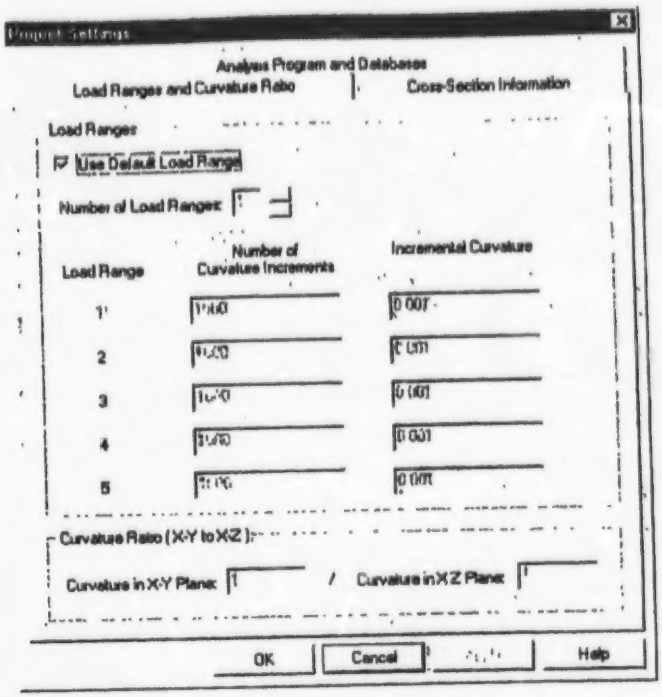


Figure 14. Project Settings Dialog (Load Ranges)

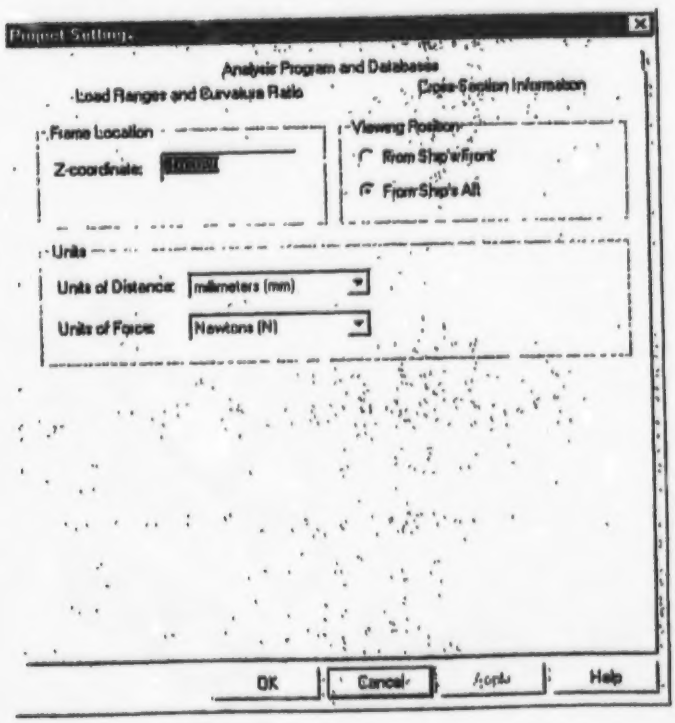


Figure 15. Project Options Dialog (Cross-section Info)

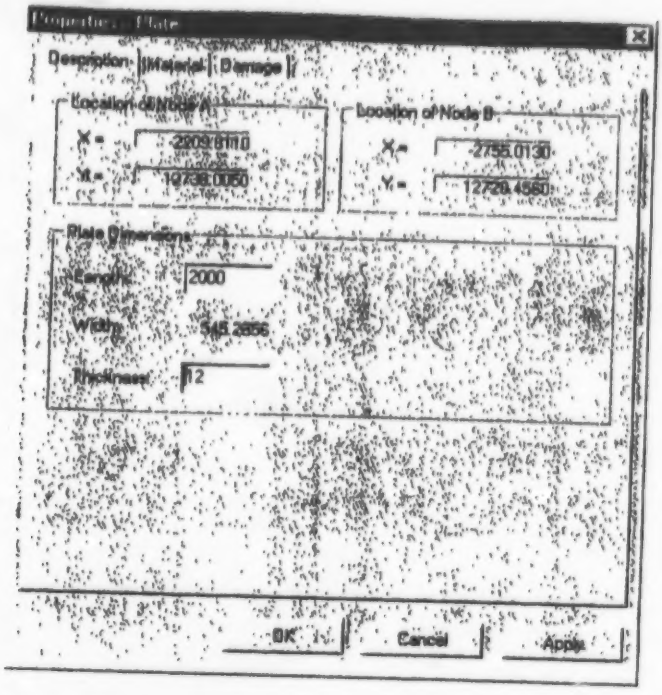


Figure 16. Plate Dialog (Description)

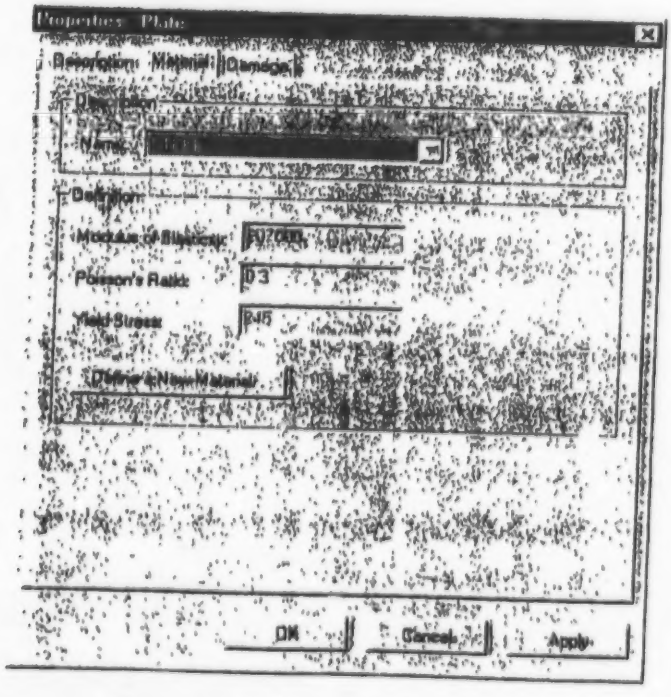


Figure 17. Plate Dialog (Material)

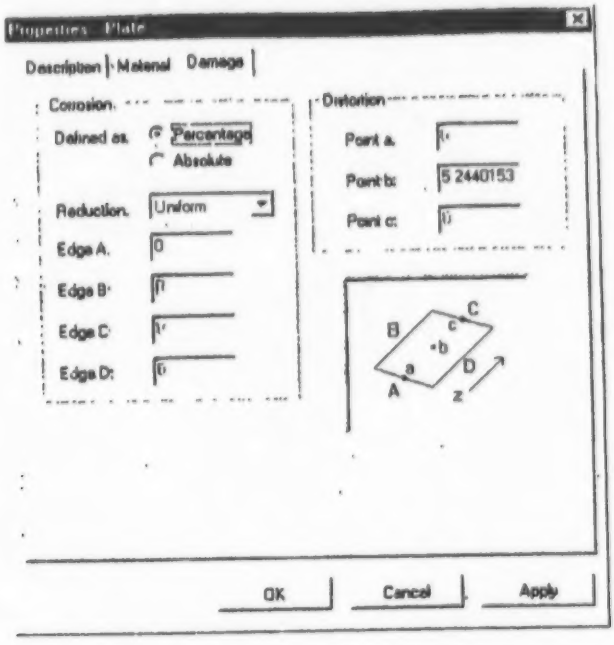


Figure 18. Plate Dialog (Damage)

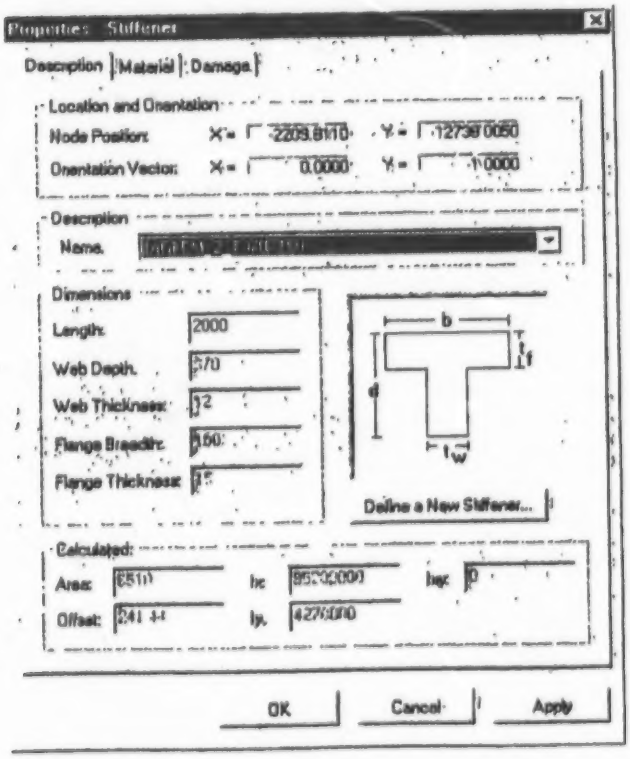


Figure 19. Stiffener Dialog (Description)

k

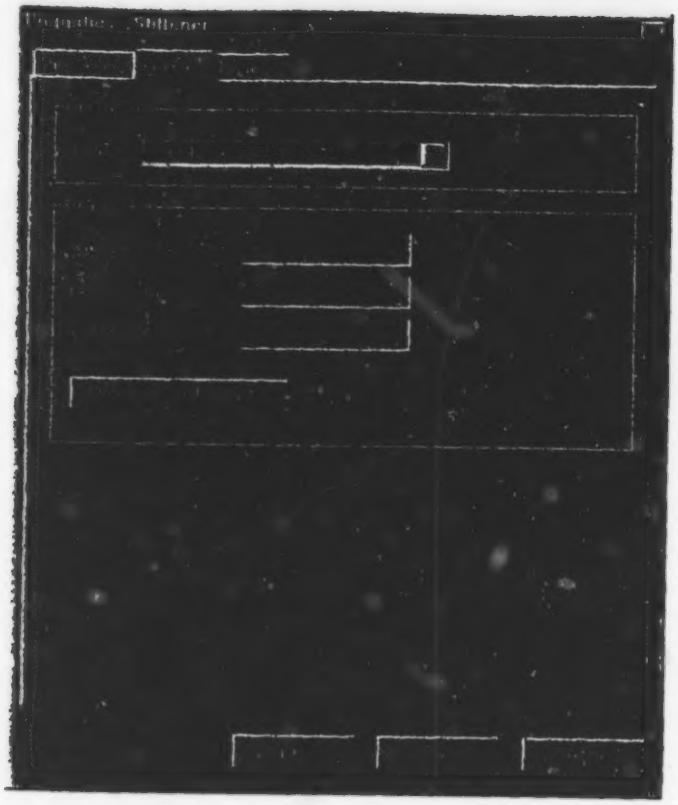


Figure 20. Stiffener Dialog (Material)

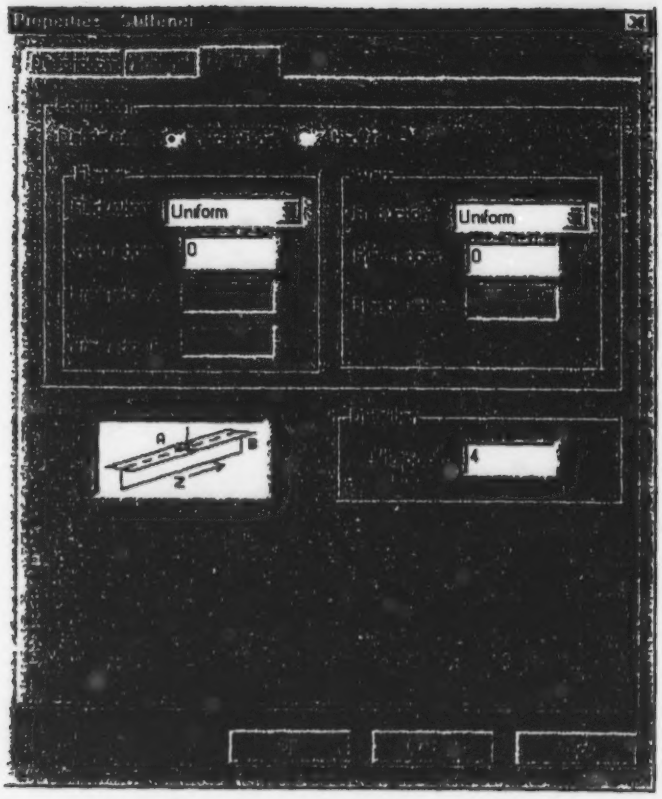


Figure 21. Stiffener Dialog (Damage)

m

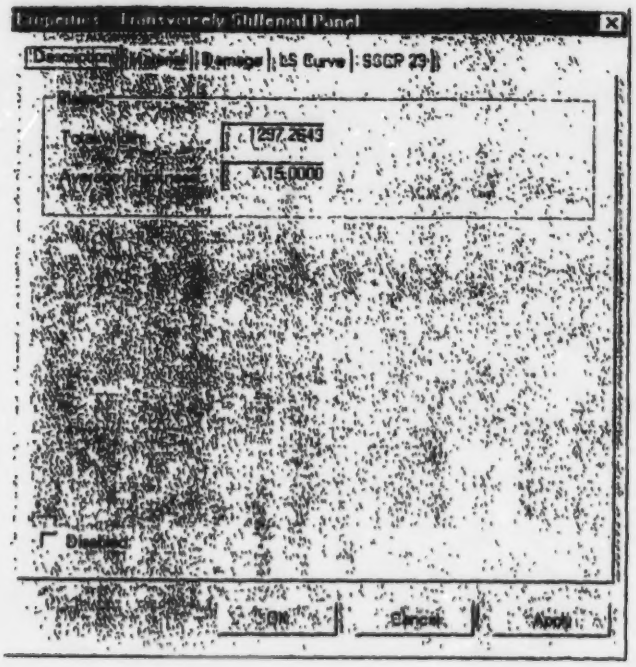


Figure 22. Transversely Stiffened Panel Dialog (Description)

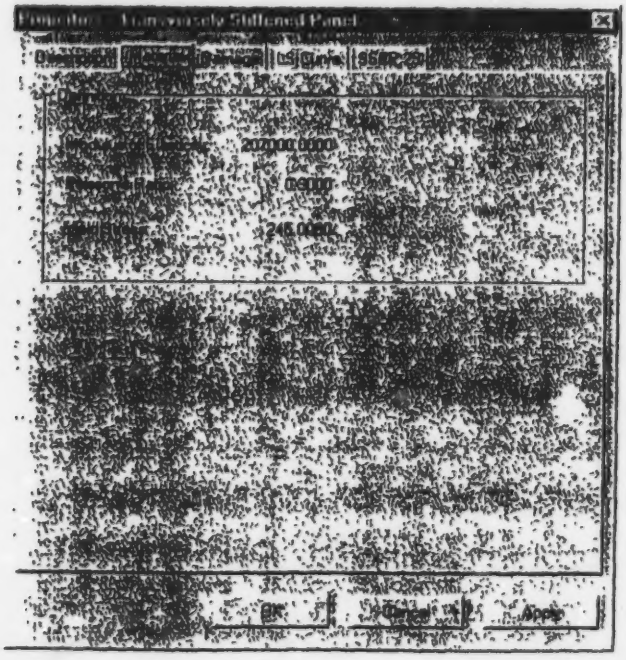


Figure 23. Transversely Stiffened Panel Dialog (Material)

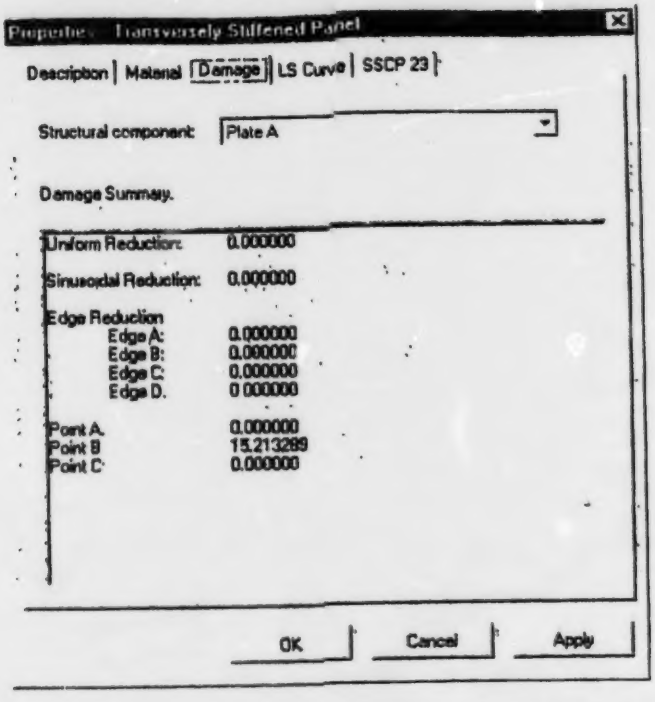


Figure 24. Transversely Stiffened Panel Dialog (Damage)

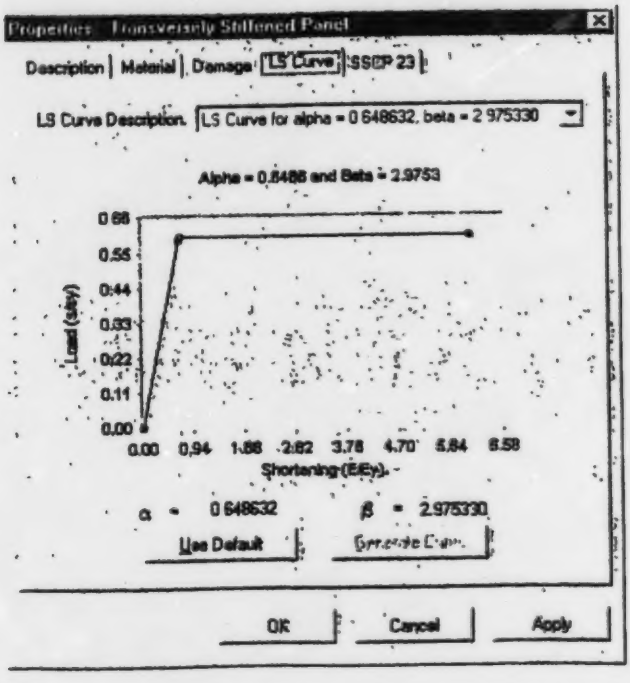


Figure 25. Transversely Stiffened Panels Dialog (LS-Curve)

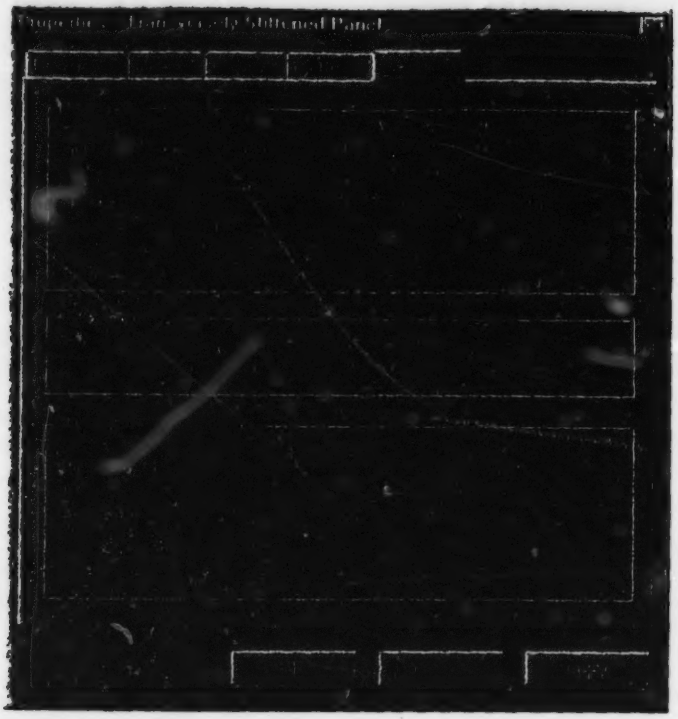


Figure 26. Transversely Stiffened Panel Dialog (SSCP23)

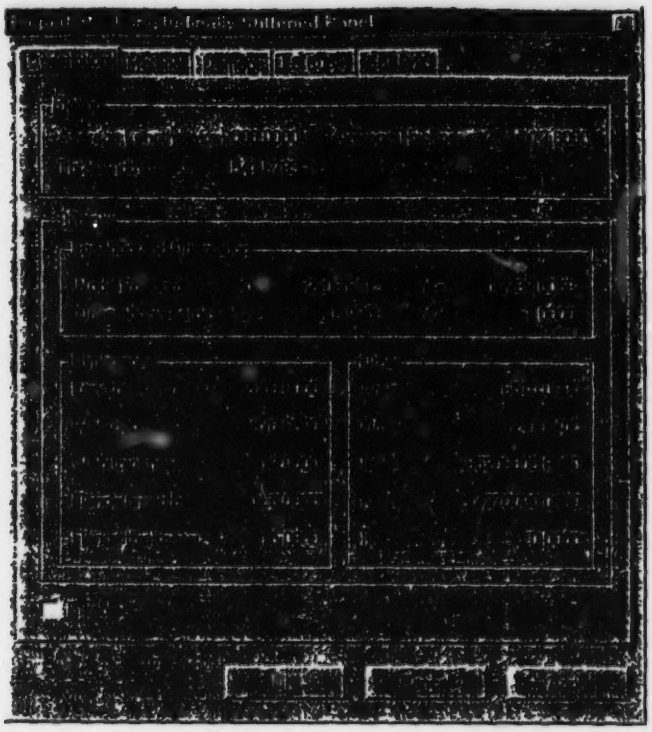


Figure 27. Longitudinally Stiffened Panel (Description)

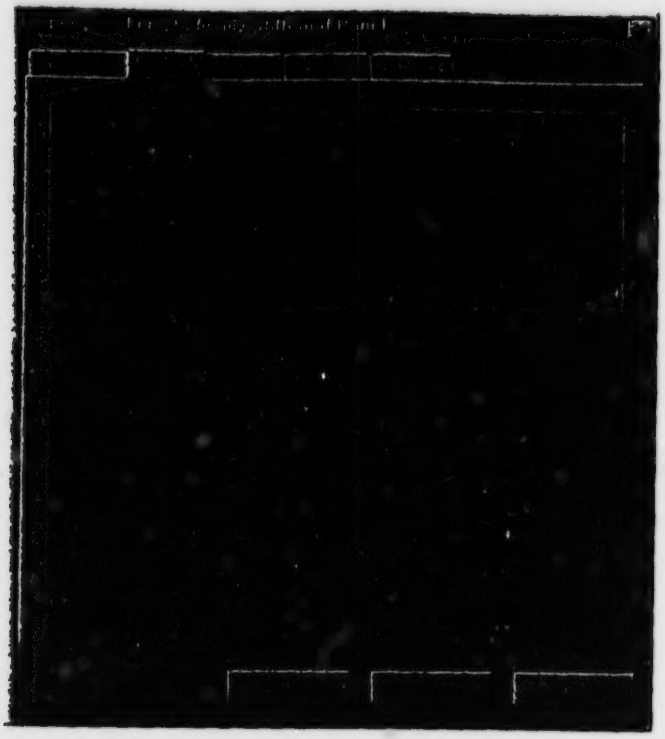


Figure 28. Longitudinally Stiffened Panel (Material)

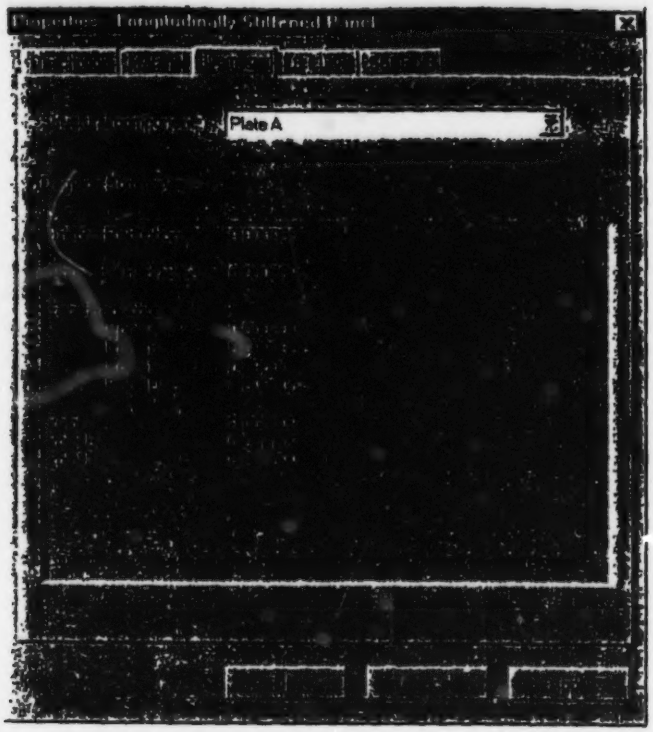


Figure 29. Longitudinally Stiffened Panel (Damage)

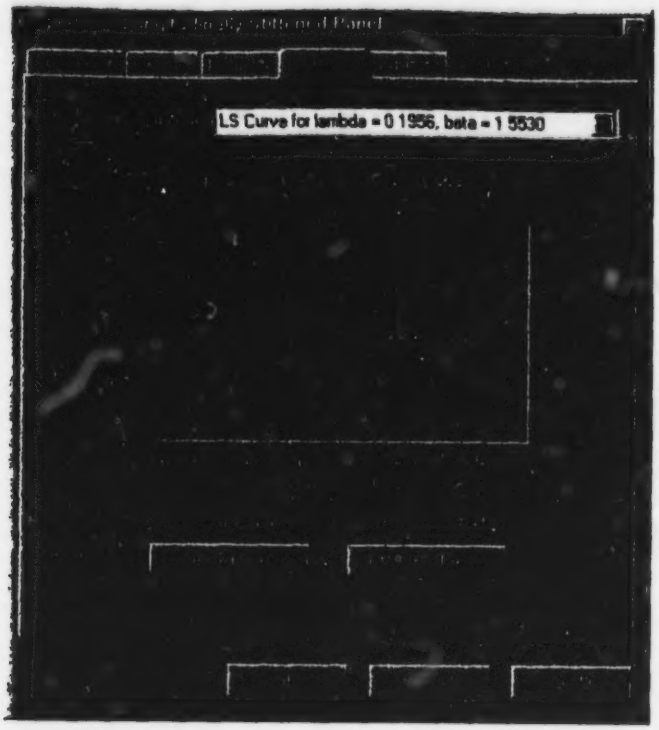


Figure 30. Longitudinally Stiffened Panel Dialog (LS-Curve)

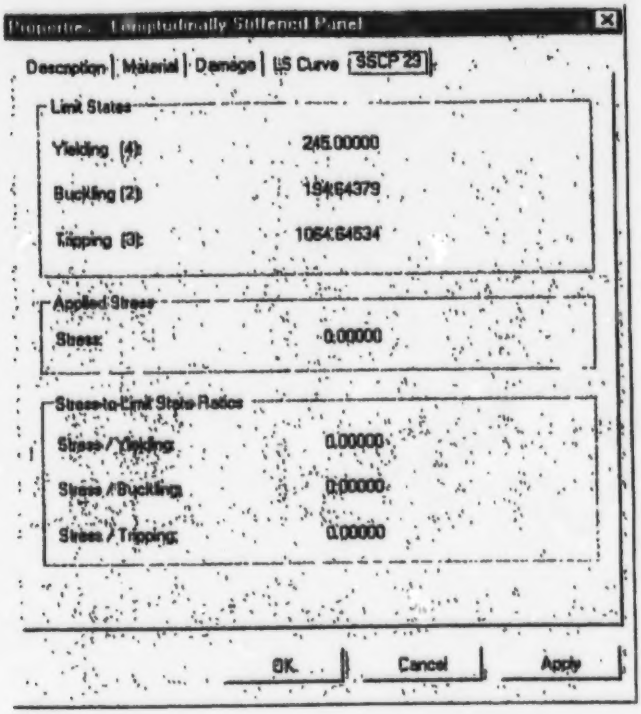


Figure 31. Longitudinally Stiffened Panels Dialog (SSCP23)

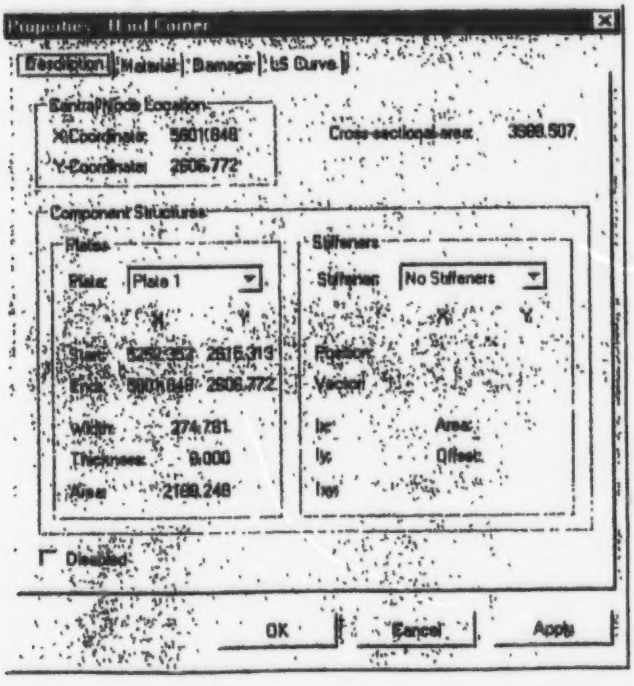


Figure 32. Hard Corner Dialog (Description)

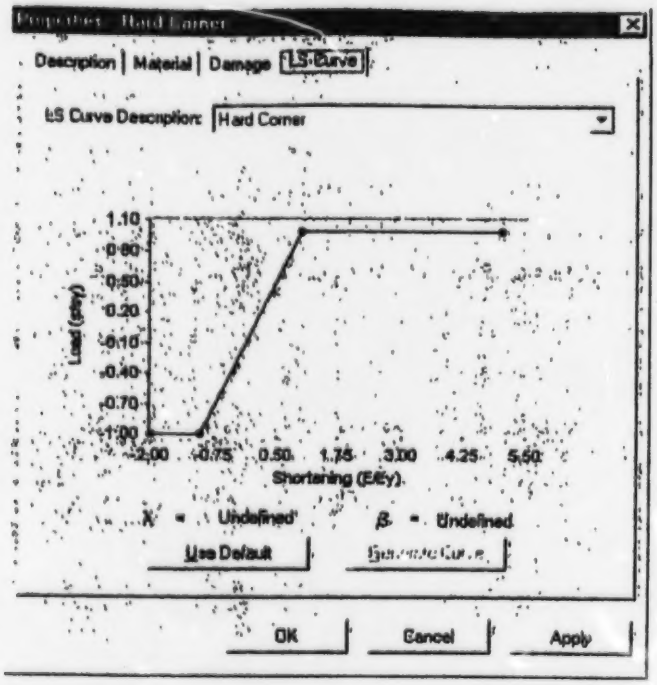


Figure 35. Hard Corner Dialog (LS-Curve)

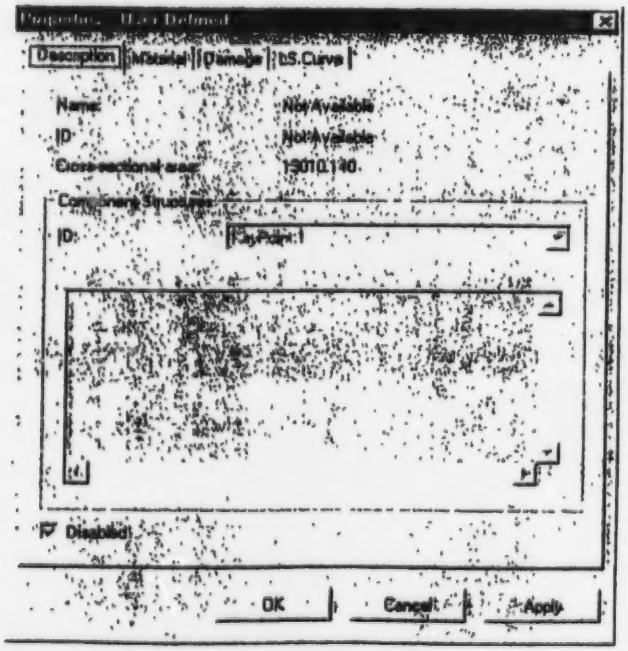


Figure 36. User Defined Structure Dialog (Description)

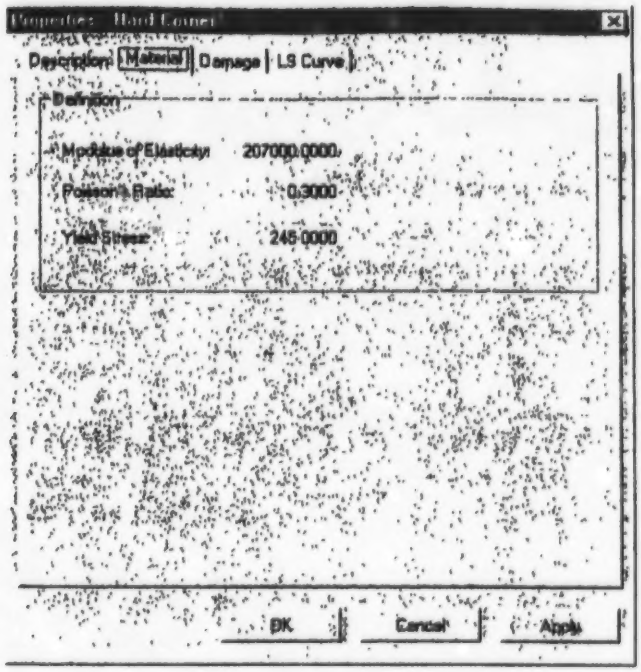


Figure 33. Hard Corner Dialog (Material)

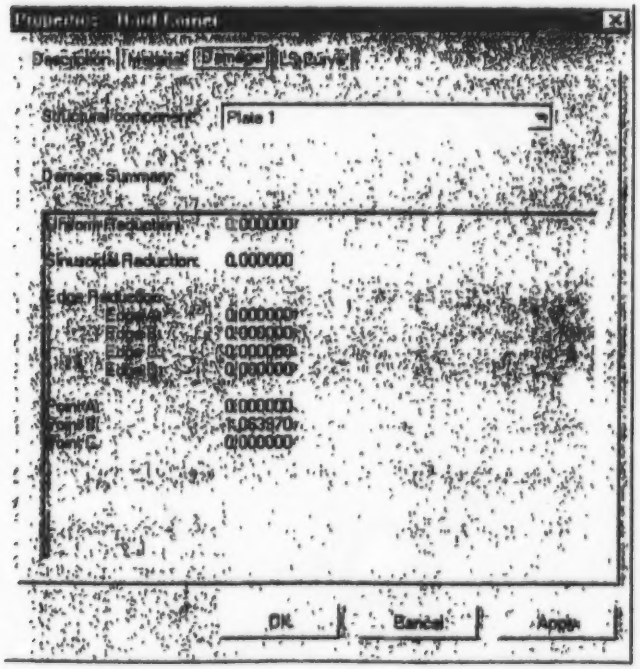


Figure 34. Hard Corner Dialog (Damage)

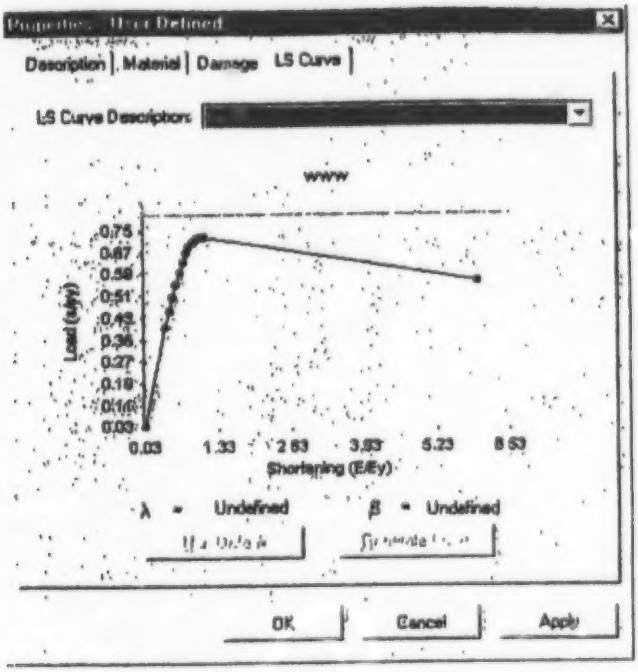


Figure 39. User Defined Structure Dialog (LS-Curve)

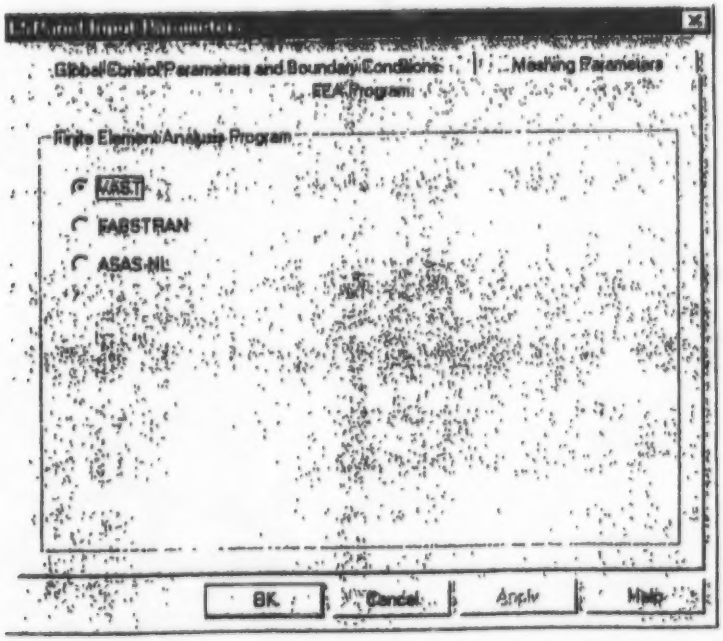


Figure 40. LSPanel Dialog (FEA Program)

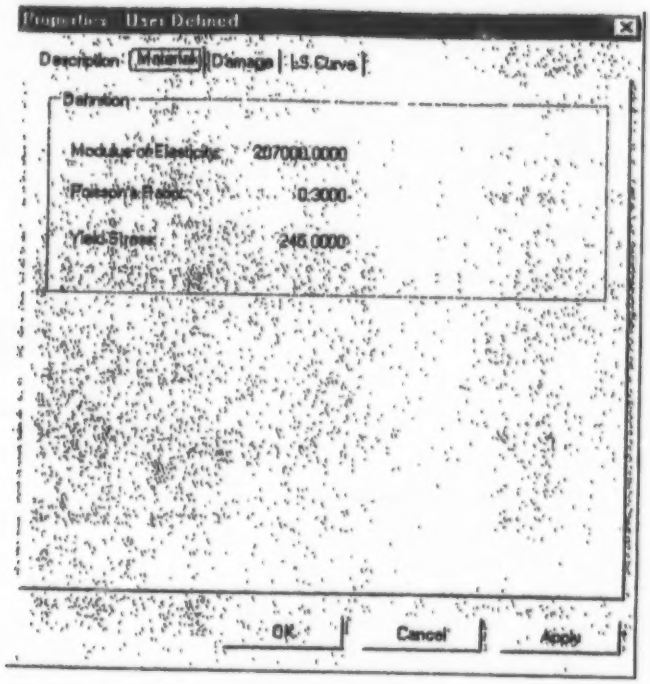


Figure 37. User Defined Structure Dialog (Material)

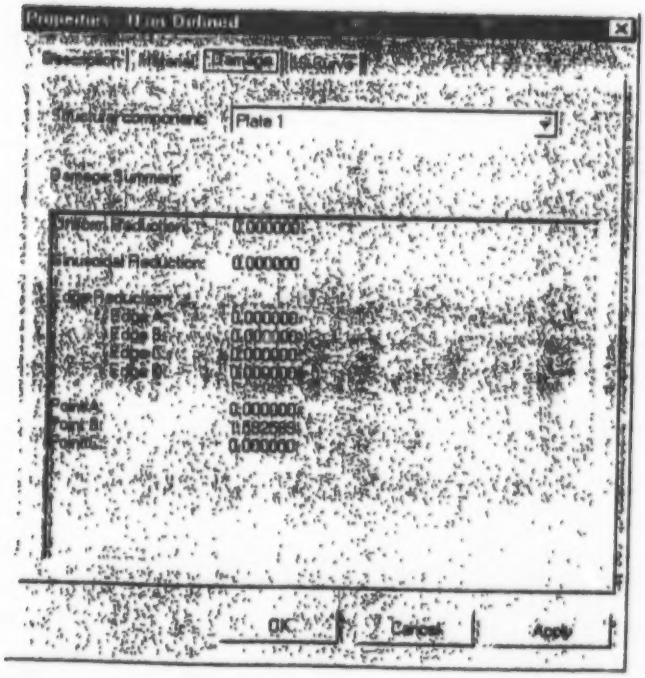


Figure 38. User Defined Structure (Damage)

y

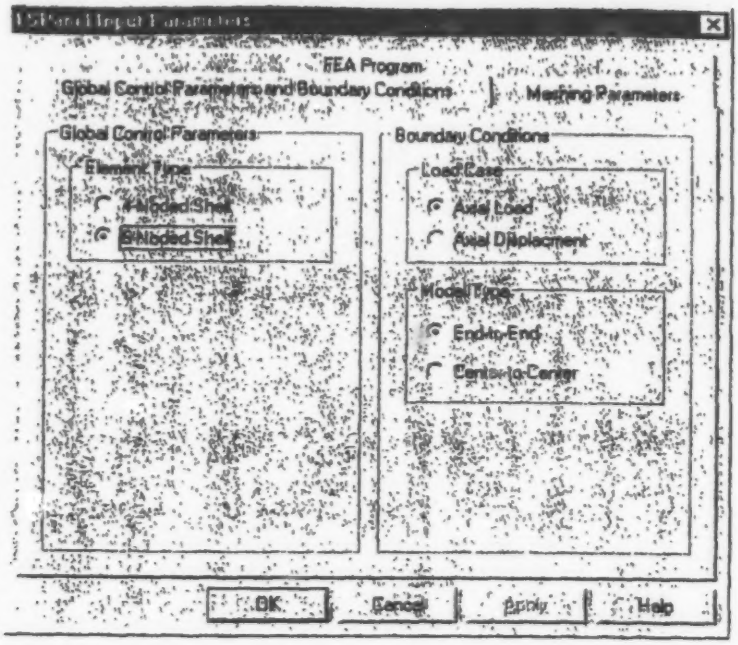


Figure 41. LSPanel Dialog (Global Control Parameters and Boundary Conditions)

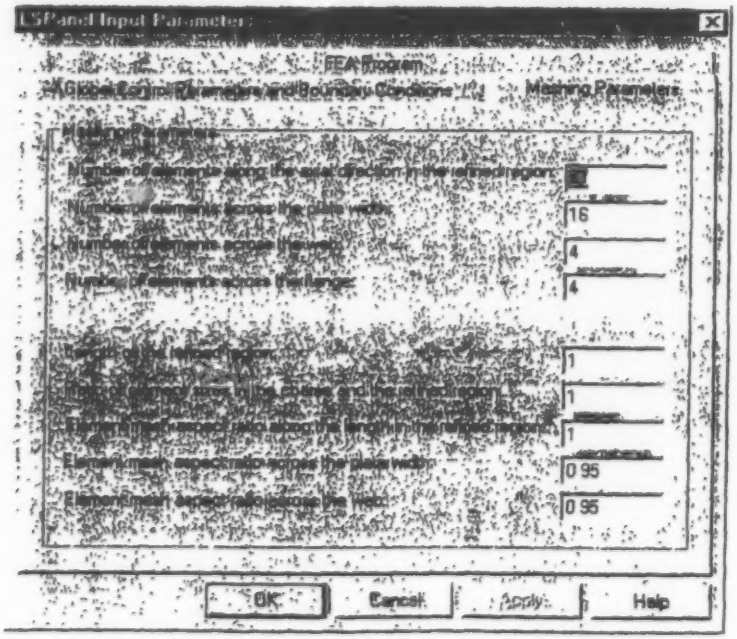


Figure 42. LSPanel Dialog (Meshing Parameters)

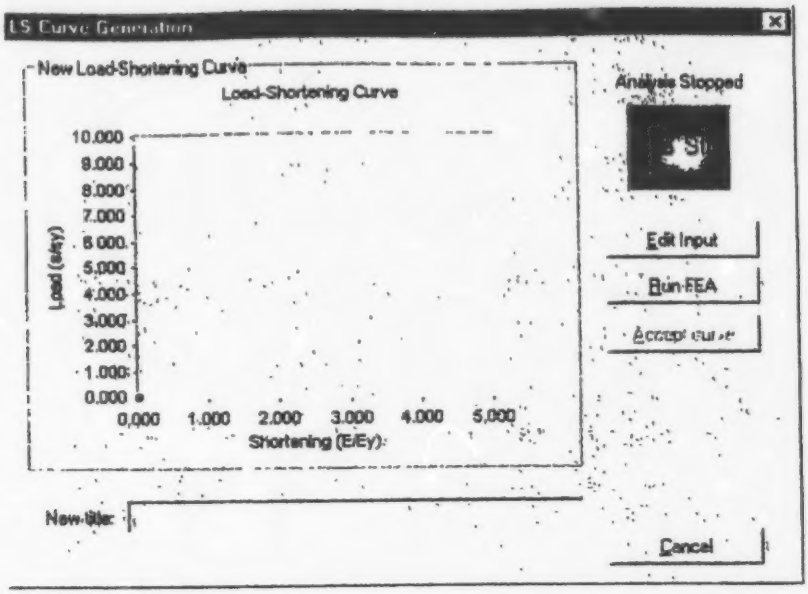


Figure 43. LS-Curve Generation Dialog (Before FEA)

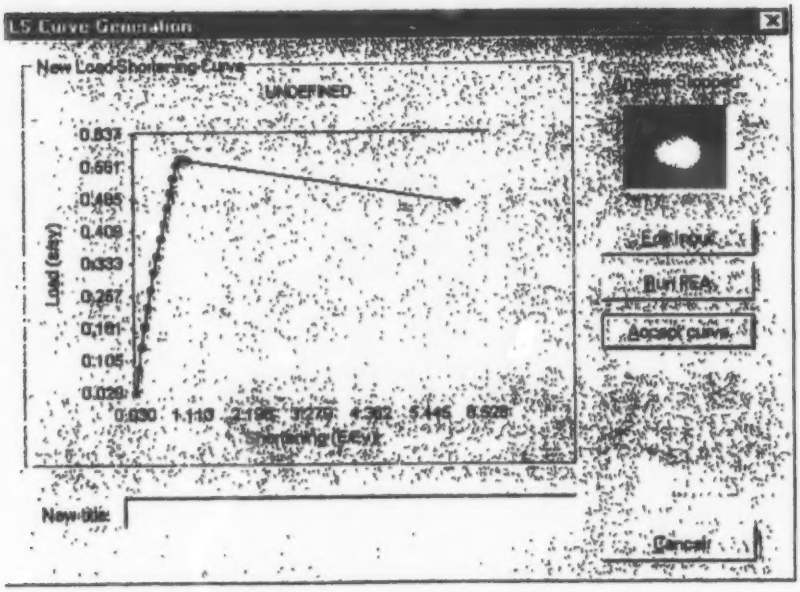


Figure 44. LS-Curve Generation Dialog (After FEA)

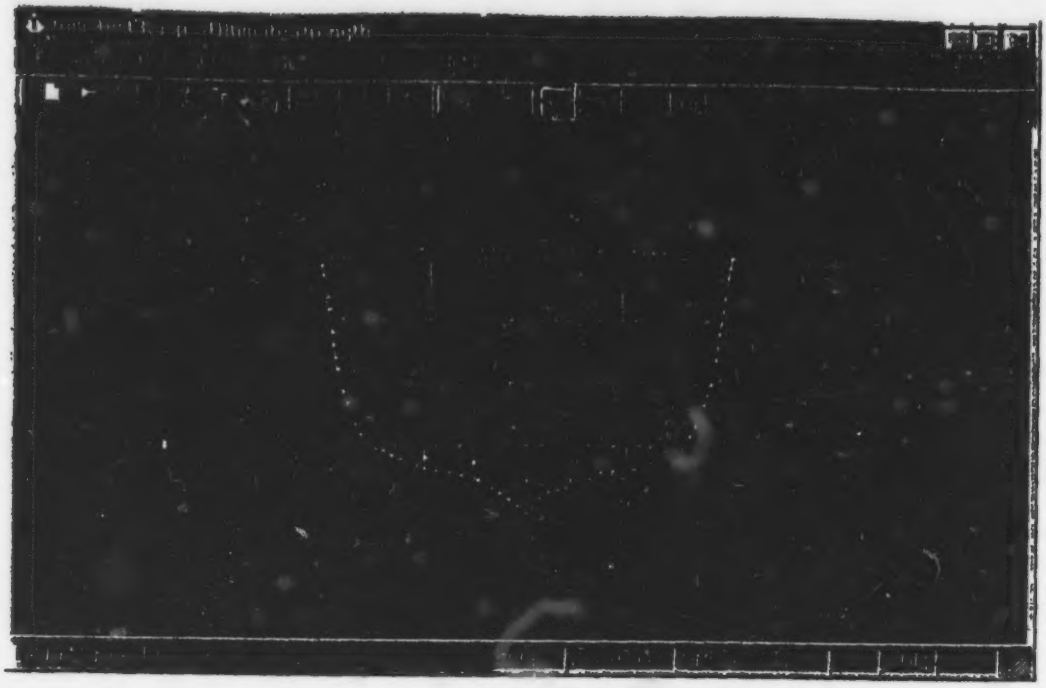


Figure 45. Finite Element Viewing Mode

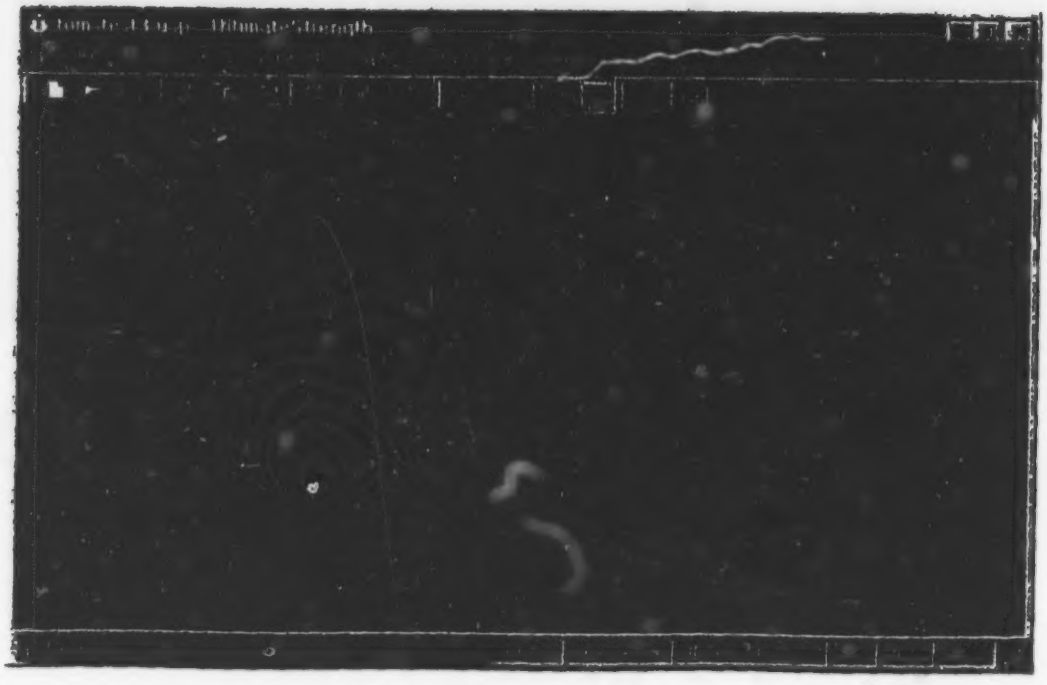


Figure 46. Structural Units Viewing Mode

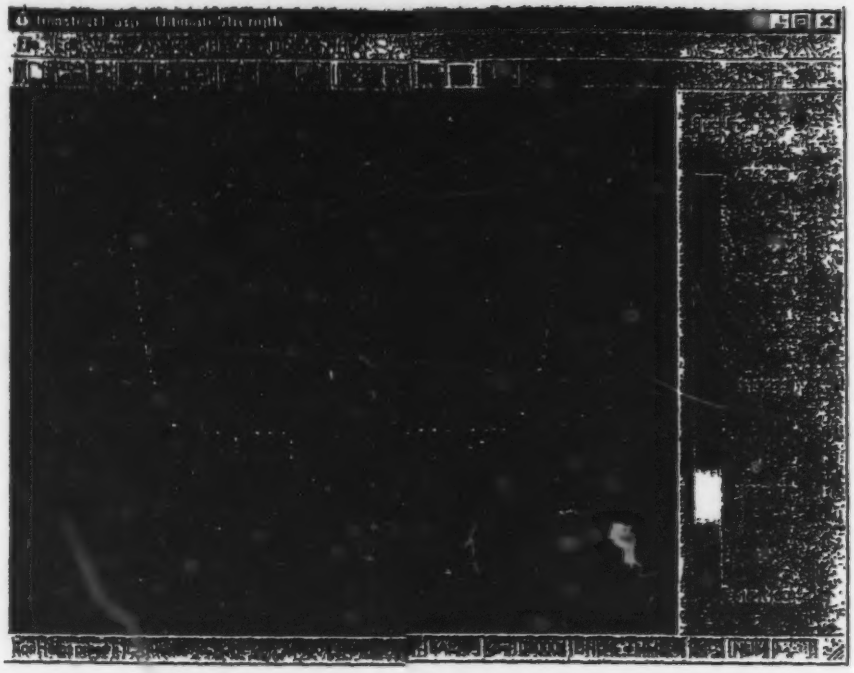


Figure 47. Graphical View of SSCP23 Results

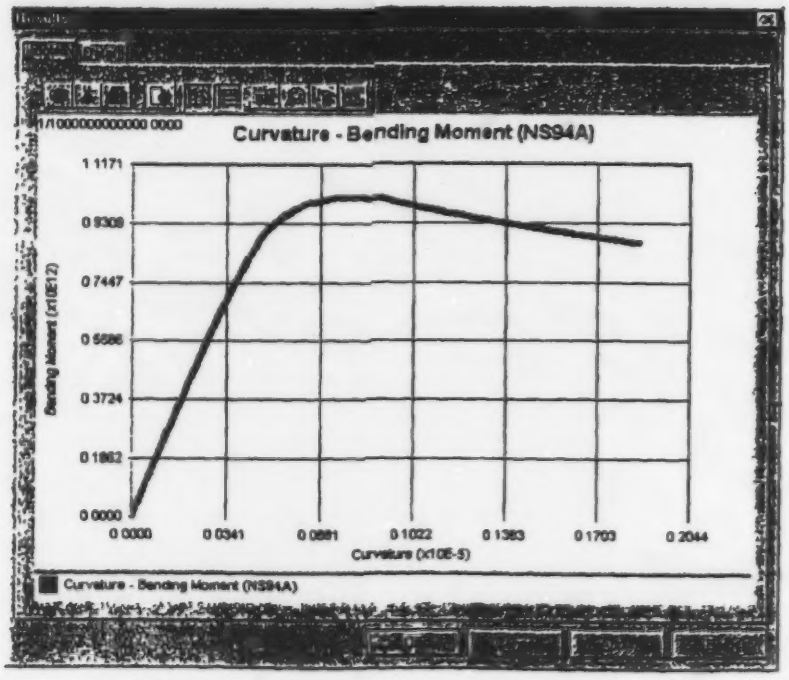


Figure 48. Results Dialog (Plot)

cc

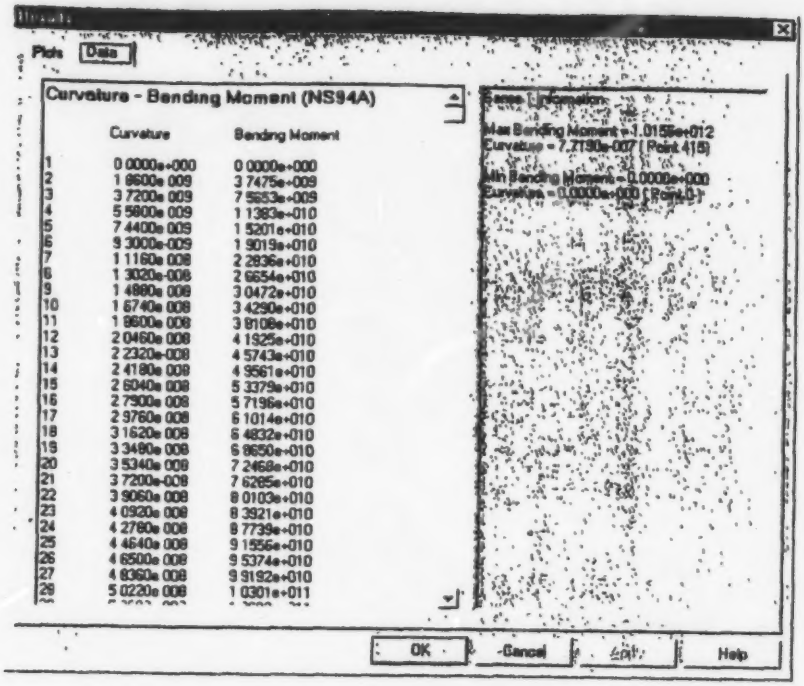


Figure 49. Results Dialog (Data)

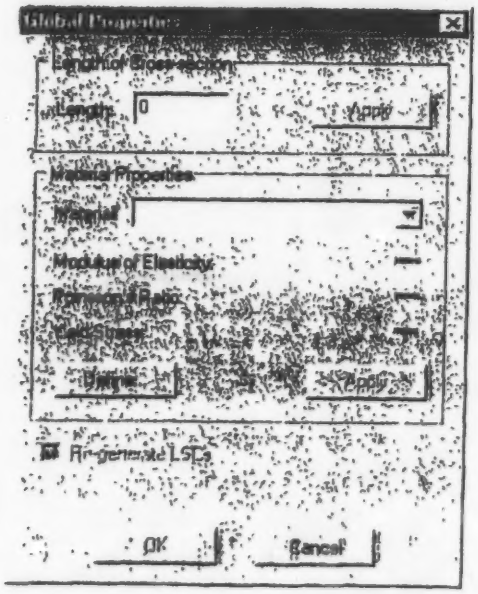


Figure 50. Global Properties Dialog

dd

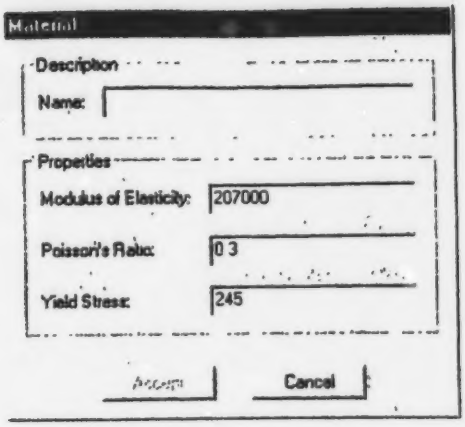


Figure 51. Material Definition Dialog

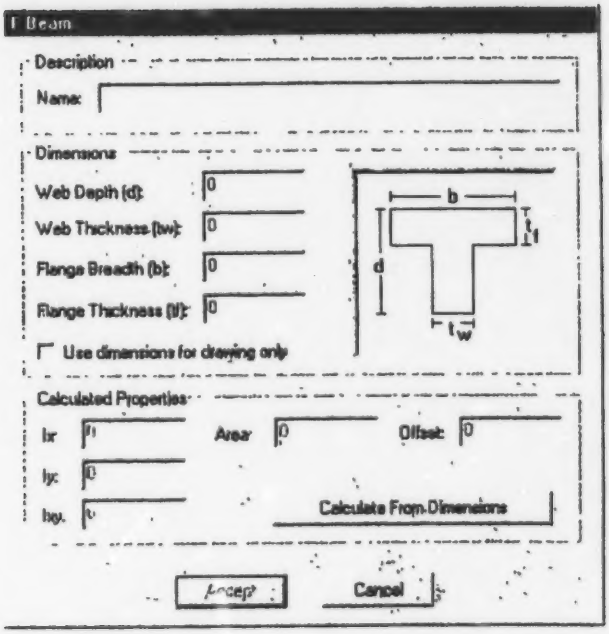


Figure 52. Stiffener Definition Dialog

ee

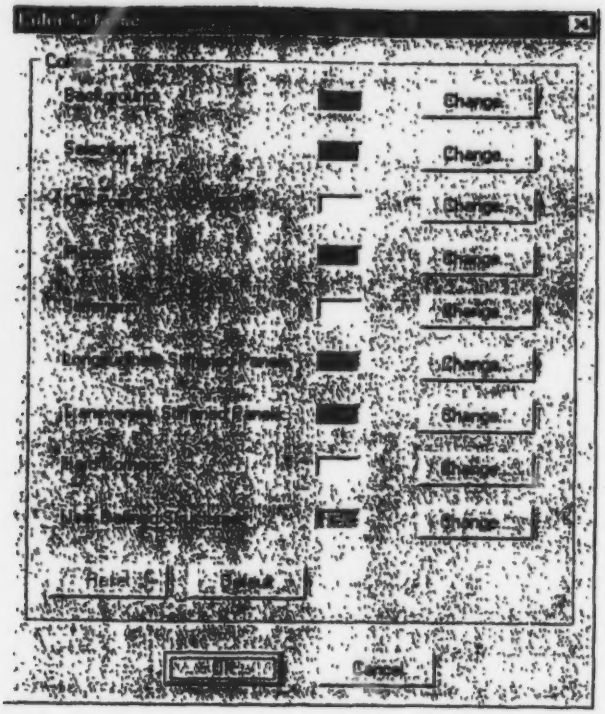


Figure 53. Color Scheme Dialog

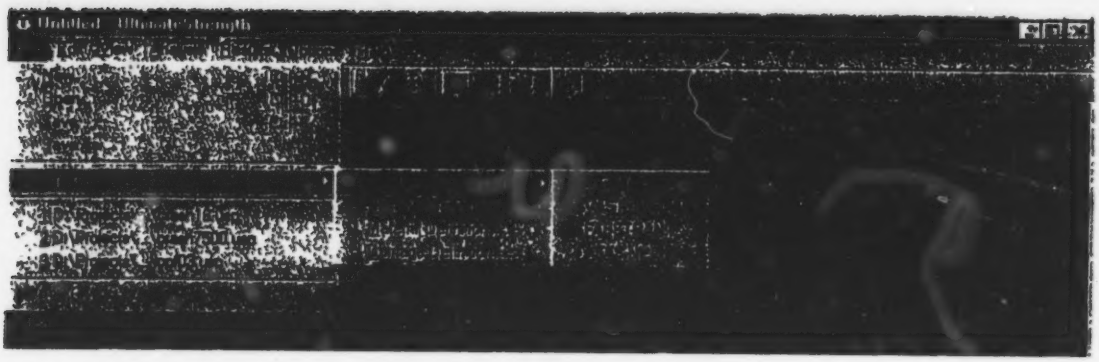


Figure 54. The File Menu

Figure 55. The Edit Menu

Figure 56. The View Menu

Figure 57. The Analysis Menu

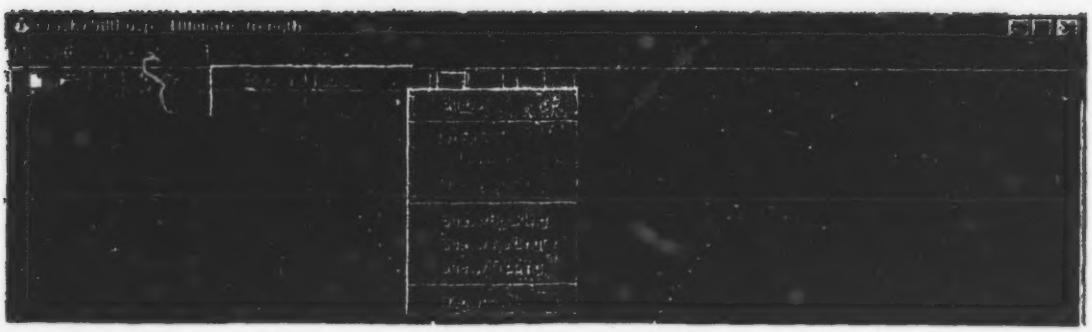


Figure 58. The Results Menu

Figure 59. The Options Menu

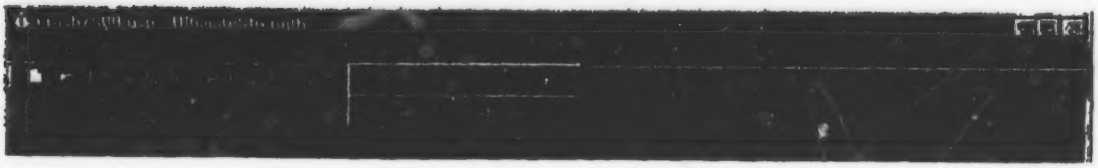


Figure 60. The Help Menu

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM
(highest classification of Title, Abstract, Keywords)

DOCUMENT CONTROL DATA

(Security classification of title, body of abstract and indexing annotation must be entered when the overall document is classified)

1. ORIGINATOR (the name and address of the organization preparing the document.. Organizations for whom the document was prepared, e.g. Establishment sponsoring a contractor's report, or tasking agency, are entered in section 8)

Martec Ltd., Suite 400, 1888 Brunswick St., Halifax, NS, B3J 3J8

2. SECURITY CLASSIFICATION (overall security classification of the document including special warning terms if applicable).

UNCLASSIFIED

3 TITLE (the complete document title as indicated on the title page. Its classification should be indicated by the appropriate abbreviation (S,C,R or U) in parentheses after the title).

Ultimate Strength Application Version 1.0 User's Manual

4. AUTHORS (Last name, first name, middle initial If military, show rank, e g. Doe, Maj. John E.)

Lichodzijewski, M., and MacAdam, T.

5 DATE OF PUBLICATION (month and year of publication of document)

March 2000

6a. NO OF PAGES (total containing information Include Annexes, Appendices, etc)

55 (approx.)

6b NO. OF REFS (total cited in document)

0

7 DESCRIPTIVE NOTES (the category of the document, e g technical report, technical note or memorandum. If appropriate, enter the type of report, e g interim, progress, summary, annual or final Give the inclusive dates when a specific reporting period is covered).

CONTRACTOR REPORT

8 SPONSORING ACTIVITY (the name of the department project office or laboratory sponsoring the research and development. Include address)

Defence Research Establishment Atlantic
PO Box 1012
Dartmouth, NS, Canada B2Y 3Z7

9a PROJECT OR GRANT NO (if appropriate, the applicable research and development project or grant number under which the document was written Please specify whether project or grant)

1.g.c.15

9b. CONTRACT NO (if appropriate, the applicable number under which the document was written)

W8477-8-0004/001/SV

10a ORIGINATOR'S DOCUMENT NUMBER (the official document number by which the document is identified by the originating activity This number must be unique to this document)

TR-00-14

10b OTHER DOCUMENT NOs (Any other numbers which may be assigned this document either by the originator or by the sponsor)

DREA CR 2000-114

11 DOCUMENT AVAILABILITY (any limitations on further dissemination of the document, other than those imposed by security classification)

(X) Unlimited distribution
( ) Defence departments and defence contractors; further distribution only as approved
( ) Defence departments and Canadian defence contractors; further distribution only as approved
( ) Government departments and agencies; further distribution only as approved
( ) Defence departments; further distribution only as approved
( ) Other (please specify):

12 DOCUMENT ANNOUNCEMENT (any limitation to the bibliographic announcement of this document. This will normally correspond to the Document Availability (11) However, where further distribution (beyond the audience specified in (11) is possible, a wider announcement audience may be selected)

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM
(highest classification of Title, Abstract, Keywords)

13. ABSTRACT (a brief and factual summary of the document It may also appear elsewhere in the body of the document itself It is highly desirable that the abstract of classified documents be unclassified Each paragraph of the abstract shall begin with an indication of the security classification of the information in the paragraph (unless the document itself is unclassified) represented as (S), (C), (R), or (U) It is not necessary to include here abstracts in both official languages unless the text is bilingual)

The document provides instructions for using the Ultimate Strength Analysis Suite (ULTSAS). This programme brings together the UK and Canadian ultimate strength analysis codes in a single software application with a common user interface and a common database. Also provided in the software is a link between ultimate strength analysis tools and the finite-element based ship structural analysis program MGDSA (MAESTRO Graphics – Detail Stress Analysis).

14. KEYWORDS, DESCRIPTORS or IDENTIFIERS (technically meaningful terms or short phrases that characterize a document and could be helpful in cataloguing the document They should be selected so that no security classification is required Identifiers, such as equipment model designation, trade name, military project code name, geographic location may also be included If possible keywords should be selected from a published thesaurus e g. Thesaurus of Engineering and Scientific Terms (TEST) and that thesaurus-identified If it not possible to select indexing terms which are Unclassified, the classification of each should be indicated as with the title)

Ship structure
Ultimate strength
Moment curvature
User's manual

UNCLASSIFIED
SECURITY CLASSIFICATION OF FORM